A CSM-CFD methodology applied to the design of a cryogenic WT model

S Russo¹, J. Kirchner², J. Seiler³, G Graumann², S Adden², N Paletta²

¹ Dream Innovation, via F. Parri 1 81030 Sant’Arpino (Caserta) – Italy
² IBK Innovation GmbH & Co. KG, Butendeichsweg 2 21129 Hamburg – Germany

serena.russo@dreaminnovation.it

Abstract. The Aero-structural coupled analysis presented in this work represents an important part of the PRODIGE project, whose main purpose is the prediction of aerodynamics and hinge moment loads of the aileron of a scale model of business jet at transonic Mach numbers and flight Reynolds number. More in details, this methodology supports the design activities of the cryogenic wind-tunnel (WT) model to be tested in the European Transonic Wind tunnel (ETW). The need to implement such a kind of analysis comes from the requirement to take into account the “pure aeroelastic effect” during experimental tests and need to understand the effect of the WT model flexibility on the control surface load prediction, prior to and in parallel with the balance design. CFD evaluations are performed on the whole model, while a static Fluid-Structure-Interface (FSI) procedure is set up and, only, applied to the wing and aileron of the model. The FSI cycle is made of two blocks: a CFD evaluation part and a CSM evaluation part. Both modules are coupled within an iterative loop, where information are exchanged between them. Two angles of incidence are taken into account, i.e. the lowest value \( \alpha = -3^\circ \) and the highest one \( \alpha = 6^\circ \). Results will be compared in terms of global aerodynamic forces and pressure coefficient distributions. Moreover the new deformed shapes derived from the application of both approaches will be compared with the original one.

1. Introduction

In the fluid dynamic analysis, a critical aspect of many engineering systems is represented by the interaction between the structure and its surrounding flow field. The interaction between fluid and flexible structure is of extreme importance in many engineering applications, due to various undesired phenomena such as fluttering, buffeting and collapsing of bridges and cooling towers, fluid-excited vibration of tall building and wind turbine blades, wind-plants interaction, as well as flutter in aircraft wings [1]-[8]. The capability to couple accurate methods able to model the physics of the several aspects of the design and their interaction is increasing the accuracy of the numerical analysis, improving as consequence the designer capability to produce products of higher performances. This is particularly true if modern design methods are considered which are strongly oriented on numerical optimization procedures.

A great number of researches has been conducted to develop methods to account for fluid-structure interaction (FSI) analysis in aircrafts components design. One of the main conclusions of these researches is that FSI analysis is highly important for the design of efficient and lightweight structure of different aircraft components especially wing, aileron as well as winglets [9][10]. Several studies focused on aero-elastic analysis of high aspect ratio wing have showed the effect of wing deformation...
on aerodynamic characteristics [11]. In [12], experimental studies were conducted on sweep forward wing to analyse the effect of wing bending and twisting on aerodynamic efficiency and the main conclusion was that wing deformation causes a reduction of in lift/drag ratio. It has been found that bending and twisting of a wing can produce a decrease in lift force of up to 27.9% [13]. Other research studies focused on static aero-elastic analysis of composite wing at various flight speeds have showed that deformation increases with increase in aircraft speed and angle of attack [14]. In [15], a study conducted to predict the effect of wing deformation in hovering on lift, drag and power coefficient has demonstrated significant differences in lift and power coefficient of wings.

The present work has been carried out in the frame of the H2020/Clean Sky 2 project called PRODIGE (grant agreement No 785436, [16]). The objective of the project is to produce experimentally accurate loads and hinge moment data at high Mach and flight Reynolds numbers. The data will be the result of an experimental test campaign performed in the European Transonic Wind (ETW) tunnel. At this aim, the consortium PRODIGE has performed the design and manufacturing of a 1/16 scale model of a business jet and of a prototype hinge moment balance for the ailerons of the wind tunnel model to be tested in cryogenic conditions. Within the PRODIGE project, the goal of this work is to present a CFD (Computational Fluid Dynamic) - CSM (Computational Structural Mechanics) procedure that has been setup during the design activity with the aim to perform aero-structural analysis on a model of swept wing in transonic conditions. This procedure allows the improving of the design of the control surfaces by predicting not only more realistic loads, but also the aerodynamic performance and the level of deformation that is expected during the experimental campaign.

Section 2 presents a description of the CFD-CSM module. Section 3 and section 4 are devoted to the definition of FE and CFD model. The results are detailed in Section 5. Section 6 is devoted to the conclusions.

2. CFD-CSM module
In this section, a brief overview of the fluid-structure interaction (FSI) scheme is presented. This scheme is applied to a wind tunnel model of business class aircraft in complete configuration, including wing, aileron, fuselage, nacelle, vertical and horizontal tail (see Figure 1). In details fuselage,nacelle, vertical and horizontal tail are considered as rigid parts, while only the wing (including the aileron) is involved in the FSI procedure. For this reason, the fluid dynamic solution is evaluated on the whole model, while the deformation is only calculated on wing and aileron. The coupling scheme for FSI analysis is illustrated in Figure 2. At time $t$, the fluid solver solves the pressure field $p(x,t)$ for flow over the exposed wetted surface of the body which is the union of two regions: 1) FSI region, which is based on deformation of wing and aileron, and 2) RIGID region, the rigid (static) region including fuselage, nacelle, vertical and horizontal tail. In the next step, the structural solver calculates the displacement field $u(x, t)$ under loading from a pressure field $p(x, t)$, where $x$ is the position vector. At every time increment $t_n$, the solvers exchange field information over multiple iterations to converge the pressure and displacement fields towards a physically compatible solution. The convergence criterion consists in the comparison between the aerodynamic forces (i.e. $C_L$, $C_D$ and $C_M$) at a certain time step with the aerodynamic forces computed at the previous time steps. If the percentage change of the aerodynamic forces is within a pre-defined tolerance, the loop is stopped. To ensure the quality of the investigation the following two alternative approaches are considered.

1. "UBS" Approach (i.e. Deformed Basic Shape) consists of following steps:
   I. Aerodynamic loads are applied to the undeformed model. This leads to a deformed model with a specific tip displacement.
   II. Afterwards, new aerodynamic loads are calculated for the deformed shape from the previous step, then these loads are being mapped back onto the undeformed shape and being applied onto the undeformed shape, which leads to a new wing deformation with a new tip displacement.
   III. Step 2 is repeated (with reference to the preceding step) until the resulting tip displacement converges.

2. "DBS" Approach (i.e. Undeformed Basic Shape) is a little different and consists of following
steps:

I. Aerodynamic loads are applied to the undeformed model. This leads to a deformed model with a specific tip displacement.

II. Afterwards, new aerodynamic loads are calculated for the deformed shape from the previous step.

III. The load differences of the two preceding steps are applied on to the deformed shape. The analysis with this new load leads to a new deformation with a new tip displacement.

IV. Step three is repeated until the converging tip displacement or load difference reaches (nearly) zero.

3. FEM model

The FE modelling utilizes TET10 elements for the individual structural parts. All connections are realized using beam elements for pins and rod elements for screws. Theses themselves are coupled to the structure via RBE2 MPCs. Overall, the model consists of roughly 490000 elements. A depiction of the full FE-model is given in Figure 3. As boundary conditions, the wing-fuselage interface flange is fully constrained.

The aerodynamic loads are represented by pressure distributions obtained from CFD which are applied to the free element faces of wing, LE-cover, aileron-covers, the aileron itself as well as the winglet. In order to avoid increased thermal stresses under cryogenic testing conditions, large deviations between thermal expansion of different materials should be avoided. This constraint results in a selection
of only two different materials, one for all structural parts (Nickel Maraging steel) and a second one (Inconel) for the connecting elements to minimize the difference of thermal expansion coefficients and have a difference of $\Delta\alpha=3.9$ $1/K$ between them.

![Figure 3. FE model](image)

4. CFD model
The Reynolds-averaged Navier-Stokes (RANS) equations are solved using implicit, upwind, second-order accurate density-based solver. The Spalart model is employed by integrating to the wall (i.e., without using wall functions) and fully turbulent flow is assumed. The problem is solved using a second-order discretization scheme initially with a CFL number of 1.0 to converge the steady-state iterative residuals by 3 orders of magnitude and a CFL number between 5 and 10 is used. After performing an iterative error analysis, the final normalized steady-state residual tolerance criteria used in this study is a 5 order of magnitude reduction (10-5).

The computational grid for 3-D flow solutions has both structured (hexahedral) and unstructured (pyramidal and tetrahedral) cells (see Figure 4 for details). The structured grid is used to capture the gradients and resolve the boundary layer near the surface of the aircraft model and of the strut. The rest of the domain has a mixture of unstructured grid blocks. A transonic Mach number and a Reynolds number of $16E6$ represent the free stream flow conditions. The most extreme conditions in terms of maximum and minimum lift in the WT are analyzed.

![Figure 4. Computational grid: (a) full model, (b) particular of the aileron](image)

5. Results
This section is devoted to a description of the results obtained by using both DBS and UBS methodologies. Figure 5 to Figure 8 and Figure 10 to Figure 12 refer respectively to $\alpha=6^\circ$ and $\alpha=-3^\circ$ conditions. Figure 5, 6, and 10 show the deformation of the wing for each iteration of the FSI procedure: the initial rigid shape is represented in grey and the red, green, blue and magenta wing shapes refer respectively to the II, III, IV and V iterations. In details Figure 5 and Figure 10(a) are based on the use
of DBS approach, while Figure 6 and Figure 10(b) are derived from the application of the UBS methodology. By comparing Figure 5 and Figure 6, it is possible to observe that the shapes at IV and V iteration (blue and magenta colors) of DBS approach can be completely overlapped; conversely, small differences are observable between the same iteration shapes according to UBS approach.

Figure 7(a) and Figure 8(a) show the trend of longitudinal and normal forces as function of the number of iterations: all the values are non-dimensionalised with the one of the first iteration, for this reason it is equal to 1. Figure 7(b) and Figure 8(b) include the percentage differences between two successive iterations. If the tolerance for the percentage change of the aerodynamic forces is fixed to 5%, it is possible to observe that it is reached at the fourth iteration. In both cases, the fifth iteration further reduce the differences between two successive iterations, by confirming the convergence of the results.

Figure 5. DBS approach (α=6°)

Figure 6. UBS approach (α=6°)

Figure 7. DBS approach (α=6°)
Figure 9(a) shows the tip displacements for the iteration steps according to DBS and UBS approaches: the final displacements are very similar even if the value is higher for UBS approach. The trends of the pressure coefficients are compared in Figure 9(b): UBS and DBS are translated with respect to the original shape because of the wing deformation. By comparing blue and green lines, it is possible to observe that both approaches give very similar results, confirming the above presented results about the deformation of the wing and the global forces. Moreover, if the pressure coefficient distribution of the deformed shapes are compared with that of the rigid case, it is possible to observe that the global trends are very similar.

Figure 9. UBS vs. DBS approach ($\alpha=6^\circ$): tip displacement and pressure coefficient trend for $y=0.65$ m

Figure 10. DBS (a) vs. UBS approach ($\alpha=-3^\circ$)

Figure 10 shows the deformation of the wing for each iteration of the FSI procedure at $\alpha=-3^\circ$: the initial rigid shape is represented in grey and the red, green and blue wing shapes refer respectively to
the II, III and IV iterations. Unlike what happens for $\alpha=-3^\circ$, this case requires one less iteration. Moreover it is possible to observe that the shapes at III and V iteration (green and blue colours) of both DBS and UBS approaches can be completely overlapped. Figure 11(a) and Figure 12(a) show the trend of longitudinal and normal forces as function of the number of iterations: all the values are non-dimensionalised with the one of the first iteration, for this reason it is equal to 1. Figure 11(b) and Figure 12(b) include the percentage differences between two successive iterations. If the tolerance for the percentage change of the aerodynamic forces is fixed to 5%, it is possible to observe that it is reached at the fourth iteration.

![Figure 11. DBS approach ($\alpha=-3^\circ$)](image1)

![Figure 12. UBS approach ($\alpha=-3^\circ$)](image2)

### 6. Conclusions

The goal of this work is to present a CFD (Computational Fluid Dynamic) - CSM (Computational Structural Mechanics) procedure that has been setup during the design activity with the aim to perform aero-structural analysis on a model of swept wing in transonic conditions. This scheme is applied to a wind tunnel model of business class aircraft in complete configuration: fuselage, nacelle, vertical and horizontal tail are considered as rigid parts, while only the wing (including the aileron) is involved in the FSI procedure. For this reason, the fluid dynamic solution is evaluated on the whole model, while the deformation is only calculated on wing and aileron. To ensure the quality of the investigation the two alternative approaches are considered: UBS and DBS methodologies. A transonic Mach number and a Reynolds number of $16E6$ represent the free stream flow conditions. The most extreme conditions in terms of maximum and minimum lift in the WT are analyzed. Both approaches produce similar results in terms of deformations and forces. In both cases the application of the FSI procedure produce lower aerodynamic loads if compared with those of the first iteration. Moreover, it is possible to observe that, if the tolerance for the percentage change of the aerodynamic forces is fixed to 5%, it is reached after 4/5 iterations depending on the aerodynamic conditions.
7. Acknowledgement
This project has received funding from the Clean Sky 2 Joint Undertaking under the European Union’s Horizon 2020 research and innovation programme under grant agreement No 785436.

8. Disclaimer
Any dissemination of results must indicate that it reflects only the author’s view and that the [Commission][Agency] is not responsible for any use that may be made of the information it contains.

References
[1] Billah, K. and R. Scanlan, Resonance, Tacoma Narrows bridge failure, and undergraduate physics textbooks. American Journal of Physics, 1991. 59(2): p. 118-24.
[2] Dooms, D., Fluid-structure interaction applied to flexible silo constructions, in Department of Civil Engineering. 2008, Katholieke Universiteit Leuven.
[3] Farhat, C., et al., Provably second-order time-accurate loosely-coupled solution algorithms for transient nonlinear computational aeroelasticity. Computer methods in applied mechanics and engineering 2006. 195(17): p. 1973-2001.
[4] Willcox, K.E., Reduced-order aerodynamic models for aeroelastic control of turbomachines. 1999, Citeseer.
[5] Jain, A., N.P. Jones, and R.H. Scanlan, Coupled flutter and buffeting analysis of long-span bridges. Journal of Structural Engineering, 1996. 122(7): p. 716-725.
[6] Yang, J.N., et al., Benchmark problem for response control of wind-excited tall buildings. Journal of Engineering Mechanics, 2004. 130(4): p. 437-446.
[7] Arrigan, J., et al., Control of flapwise vibrations in wind turbine blades using semi-active tuned mass dampers. Structural Control and Health Monitoring, 2011. 18(8): p. 840-851.
[8] De Langre, E., Effects of wind on plants. Annu. Rev. Fluid Mech., 2008. 40: p. 141-168.
[9] Wan, Z. and C. Yang, Aeroelastic optimization of a high-aspect-ratio composite wing. Fuhe Cailiao Xuebao (Acta Mater. Compos. Sin.), 2005. 22(3): p. 145-149.
[10] SI, L., H.-p. Wang, and C.-c. Gong, Investigation of Effects of Winglets on Wing’ s Flutter Behavior [J]. Aeronautical Computing Technique, 2009. 4: p. 023.
[11] Changchuan, X., W. Zhigang, and Y. Chao, Aeroelastic analysis of flexible large aspect ratio wing. JOURNAL-BEIJING UNIVERSITY OF AERONAUTICS AND ASTRONAUTICS, 2003. 29(12; ISSU 130): p. 1087-1090.
[12] Hua, M.T.M.D.Z., Aerodynamic characteristic analysis of high-aspect ratio elastic wing [J]. Journal of Beijing University of Aeronautics and Astronautics, 2007. 7: p. 009.
[13] Shi, A.-M., Y.-N. Yang, and G. Wang, Investigations of characteristics of static aeroelasticity for elastic wing in transonic flow [J]. Engineering Mechanics, 2006. 23(5): p. 173-176.
[14] Huo, S.-h., et al., Effects of static aeroelasticity on composite wing characteristics under different flight attitudes. Journal of Central South University, 2013. 20: p. 312-317.
[15] Smith, M.J., D.H. Hodges, and C.E. S. Cesnik, Evaluation of computational algorithms suitable for fluid-structure interactions. Journal of Aircraft, 2000. 37(2): p. 282-294.
[16] https://cordis.europa.eu/project/id/785436