Coupling of Fluid, Structures and Waves in Aeronautics 2001
Melbourne, Australia, December 3-6, 2001
Notes on Numerical Fluid Mechanics, volume 85
Springer-Verlag – 2003

Status and Future Challenges of CFD in a Coupled Simulation Environment for Aircraft Design

F. CHALOT, T. FANION, M. MALLET, M. RAVACHOL and G. ROGE
Dassault Aviation
78 quai Dassault
92214 Saint Cloud - France

Summary

The state of the art of Computational Fluid Dynamics and the axis of improvements are described. The issue of flutter prediction is addressed first : the use of linearized Euler solvers for transonic flutter is explained. Recent advances in optimum aerodynamic shape design are presented next , the results demonstrate the applicability of optimization based on the Euler equations and open the way to multidisciplinary optimum design. Finally, the use of Large Eddy Simulation for accurate turbulent flow simulation is illustrated.

1 Introduction

Computational Fluid Dynamics has reached a high level of maturity and has a considerable impact on aircraft design. This has been foreseen for some time [9] and described in a number of publications for example in [3].

The presentation will illustrate the present capability of CFD and also identify the areas where development efforts are made. Coupling with structural analysis for the prediction of flutter will be presented first : this is an example of a multi - physics simulation for a critical design issue for aircraft. Issues related to design optimization will be discussed next, a lot of progress is made in this area to increase design turn around capabilities. Finally, we will address the question of the accuracy of CFD simulations: this will be illustrated by the problem of turbulence modeling.

2 Fluid Structure Interaction

Multi physics simulation is an area of active development and fluid / structure interaction is typical example. For aircraft design, the main objective is to predict flutter. Flutter is the resonant interaction between unsteady aerodynamic loads and the elastic deformation of the aircraft structure. Flutter is a dangerous phenomena that can lead to the rapid loss of an aircraft. State of the art techniques rely on simplified linear numerical models for aerodynamics. This approach is fast and well calibrated (we emphasize the high number of configurations that must be analysed). However, when nonlinear flow behavior is present, these methods cannot be applied. Emerging techniques consider more complex flow models.

Two approaches can be considered. The first one relies on the coupling of non-linear unsteady CFD and unsteady structural dynamics codes. Significant effort has been devoted to the development of this approach. Critical numerical ingredients have been identified, most notably the need to use efficient staggered schemes and to fulfil the geometric conservation law criteria. This is described for example in [8]. This approach has been implemented and demonstrated in our computational environment SOUPLE. A typical result is presented in [3]. This approach is very general. It is however CPU intensive and this restricts its present application to validation study and to the analysis of the most critical and complex configurations.

An alternate approach is based on the solution of the linearized Euler equations in the frequency domain. The procedure includes the following steps. First a structural displacement basis is considered. For example a modal basis can be selected. The Generalized Aerodynamic Force matrix *GAF(f)* is computed next. For a given frequency, *GAF(f)(i,j)* is the aerodynamic pressure force associated to nodal displacement *i* when the structure oscillates along mode *j* at frequency *f*. Matrix *GAF* is obtained by solving the linearized Euler equations for each mode and a number of frequencies over a selected frequency range. Typically, the number of modes considered is between 4 and 10 and the number of frequencies around 10. Between two frequencies, linear interpolation is performed. This step requires the solution of (number of modes)X(number of frequencies) complex linear systems. This is the most CPU intensive step of the procedure, however its cost is still low.

Once matrix *GAF* is computed, the complete dynamic behaviour of the coupled system can be explored easily. The classical approach uses the ‘p-k’ method which is based on a loop over the velocity. Starting at zero velocity, for each velocity increment, the frequency and damping associated to each mode is computed. Flutter speed is reached as soon as negative damping is obtained.

The linearized Euler solver was developed on the basis of our industrial Euler code EUGENIE. It is an unstructured solver. A modified Lax Wendroff numerical scheme is selected. A 1st order Steger Warming version can also be used. Automatic differentiation was used to contribute to the generation of the code for the linearized operator. A software tool called *Odyssée* developed by INRIA was used. Significant efforts have been devoted to achieve a high level of efficiency. The GMRES linear solver was selected and incomplete LDU preconditionning is performed. Parallel implementation on distributed memory architecture has also been programmed. A detailed description can be found in [7].

The method was validated using the well known AGARD 445.6 wing test case. The experimental investigation was performed at NASA Langley transonic tunnel.

The wing has an aspect ratio of 4, a taper ratio of 0.6, the ¼ cord sweep angle is 45 degrees. The wing profile in streamwise direction is a NACA65A004. Data is available for a wide Mach number range from 0.338 to 1.141. The first four vibration modes are illustrated on Figure 1, the mesh used for the aerodynamic computations is available on Figure 2. A representative flutter diagram is presented on Figure 3. For three modes, the frequency and damping are obtained as a function of velocity. Flutter occurs when negative damping is predicted. The transonic dip phenomena is illustrated in Figure 4 : the flutter speed index is plotted as a function of the Mach number. A nonlinear behaviour can be observed through the transonic regime. We can verify that the linearized Euler based method can predict this phenomena correctly.

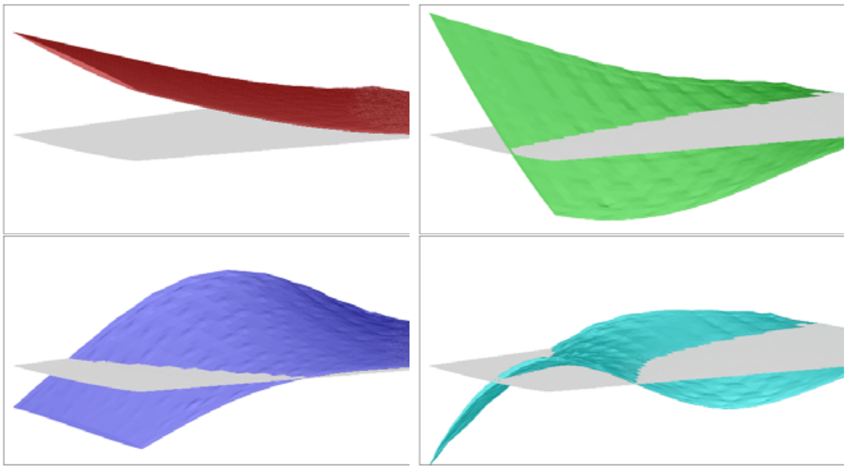

Figure 1 Vibration modes 1 to 4 - AGARD wing

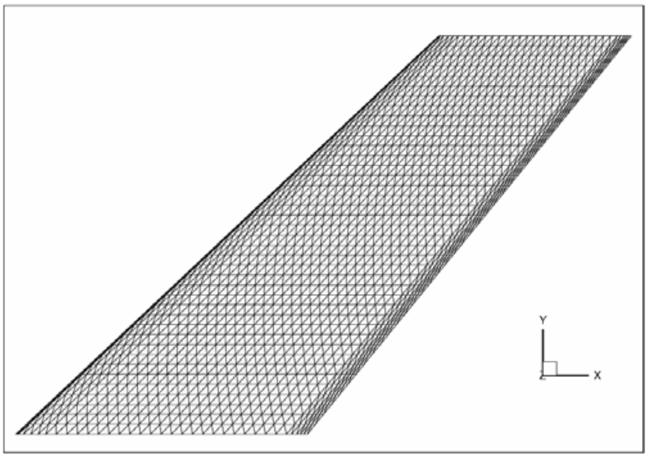

Figure 2 Mesh for Euler computation - AGARD wing

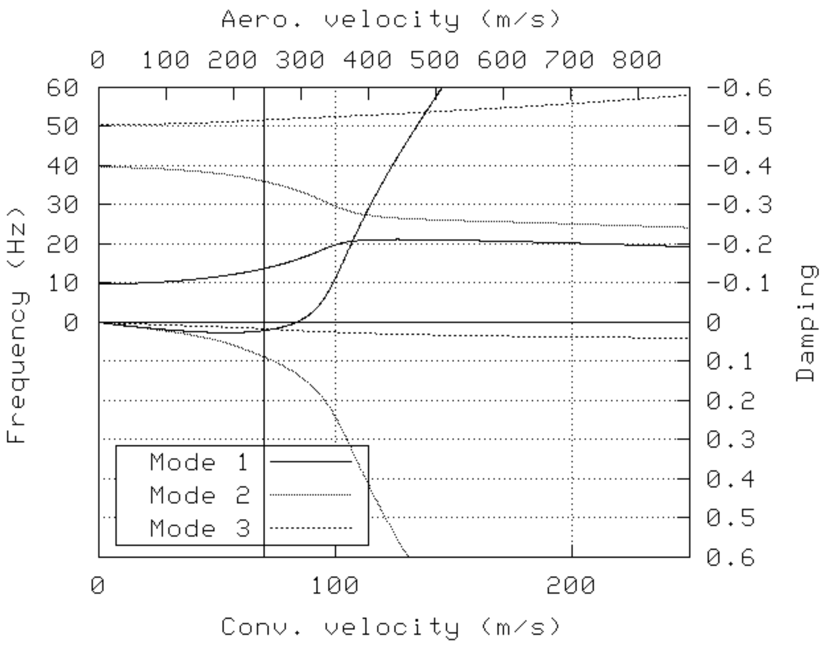

Figure 3 Flutter diagram (frequency and damping vs velocity for three modes) - AGARD wing

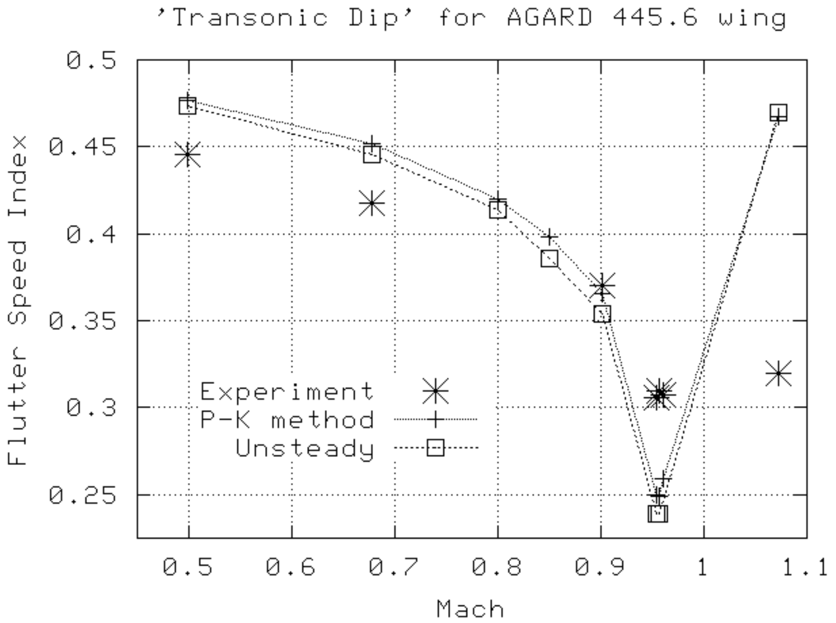

Figure 4 Transonic dip - Flutter speed vs Mach number - AGARD wing

3 Optimum Shape Design

Shape optimization is a very promising tool to improve and accelerate future aircraft design. Optimum shape design can be considered at two levels. At the early design stage, simplified models yield sufficient precision and strong multidisciplinary coupling is performed. When detailed design is reached, more accurate models are required for each discipline and multidisciplinary design is more difficult. One bottleneck that has delayed the impact of multidisciplinary optimization is the high cost of aerodynamic optimization. Considerable effort has been devoted to make progress in this field [6].

The general formulation of a shape optimization problem has been introduced by many authors ([14], [11]) and the key role played by the adjoint equation to obtain the design sensitivities in order to compute efficiently the gradient components of the cost function has been described. The adjoint equations themselves are formed as a system of partial differential equations. These can be discretized and solved in the same manner as the original flow equations. This approach is often called a continuous sensitivity analysis. We have selected a different approach, called a discrete sensitivity analysis which consists in applying the control theory to the discrete equations. The software to iteratively solve the adjoint equation was developed with the help of the *Odyssee* automatic code differentiation tool. The original code is the same 3D parallel unstructured Euler code EUGENIE mentioned in section 2.

A typical example of the design capability that has been reached is illustrated for the design of the transonic wing of a business jet. A complete configuration is considered (wing, fuselage, nacelle, pylon, empennage). The configuration is presented in Figure 5. An inverse design problem is considered: the objective is to reach a target pressure over the wing. The target pressure is defined to reduce the wave drag associated to the shock. The complete aircraft lift is constrained to be equal to a given value. CAD variables are used to define the shape parameters : this includes location, tangent and curvature parameters at control points. 96 shape variables are considered. The result of the optimization process is illustrated in Figure 6. The shock has been considerably reduced. The shape modification is illustrated for one wing section on Figure 7. The cost of the optimization process is only 6 times the cost of the solution of the Euler equations in this configuration. This is extremely efficient if we consider that 15 design iterations have been performed and 96 design variables are updated.

One issue associated to optimum design is non convex optimization where gradient based methods cannot be used alone. In this case, stochastic optimization should be used. See for example [13].

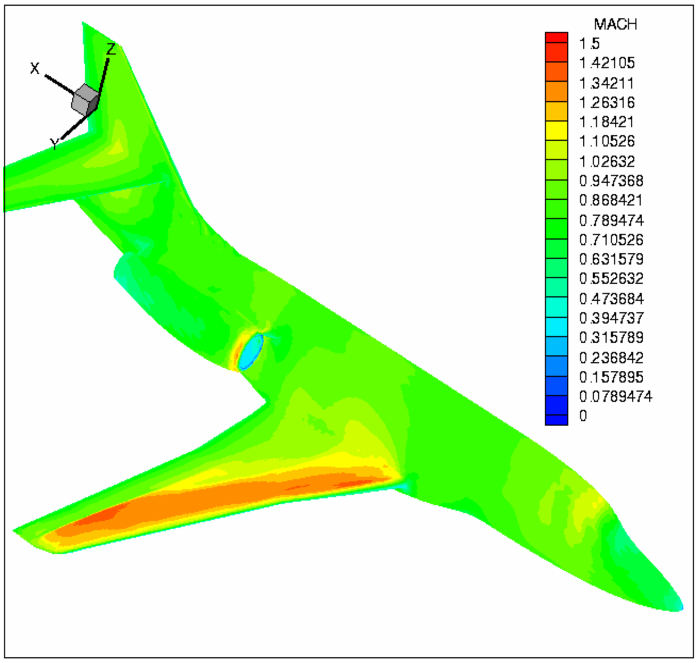

Figure 5 Optimum shape design - Falcon configuration

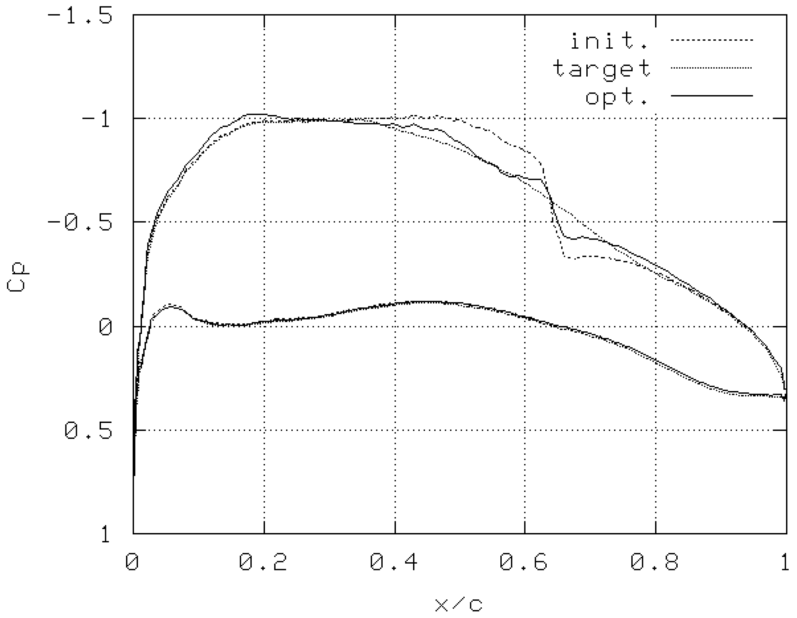

Figure 6 Optimum shape design - Pressure along wing section

Coupling of Fluid, Structures and Waves in Aeronautics 2001
Melbourne, Australia, December 3-6, 2001
Notes on Numerical Fluid Mechanics, volume 85
Springer-Verlag – 2003

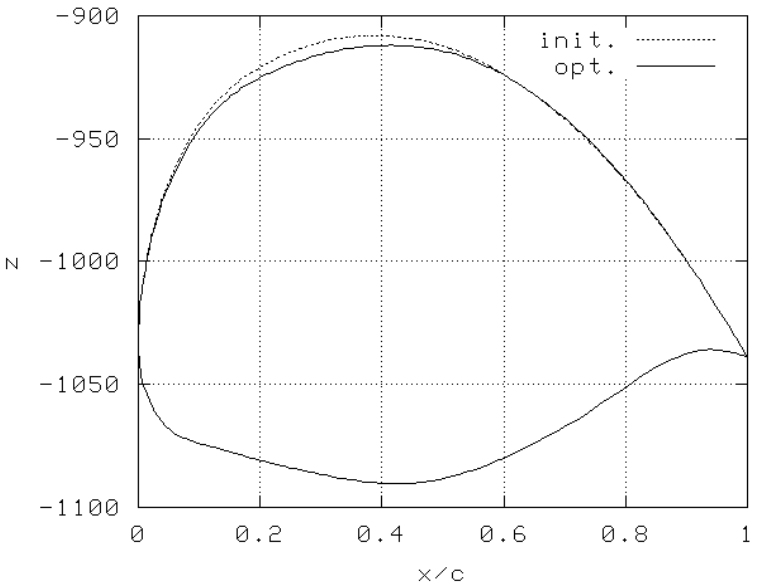

Figure 7 Optimum shape design - Initial and optimized shapes

4 Turbulence Modeling

The impact of CFD for aircraft design is still limited for many applications by a lack of accuracy. This problem can often be traced to inadequate turbulence modeling for complex flow configurations. Examples of such situations include flows with large separations like high angle of attack flows over delta wings or air inlets, flows configurations with active or passive flow control devices, afterbody flows.

One way to improve turbulence models is to consider advanced Reynolds averaged models. Another approach is based on the Large Eddy Simulation model. This approach will not replace simulation based on the solution of the Reynolds Average Navier Stokes equations with models like the (k,ε) model [15]. However it can be used for special challenging applications and for validation.

The work presented in [2] describes how our industrial unstructured NS code *AETHER* was modified to gain an LES capability (see [1] and [10] for a description of the numerical features of *AETHER*). Modifications includes improvement of the time stepping scheme, modification of the boundary condition treatment and implementation of a selective Smagorinsky model [5]. LES was then applied to the study of the mixing enhancement of a compressible mixing layer. This study was presented in [4]. Mixing enhancement is obtained by a small jet blowing perpendicular to the plane mixing layer. We present below the result of an extension of this study : the influence of jet pulsation was studied numerically. The instantaneous temperature field in a plane perpendicular to the mixing layer are presented in Figure 8 for both cases (continuous and pulsed jet). A comparison of the average velocity profile downstream of the injection device is available on Figure 9. Improved mixing is obtained with a pulsed jet at this location not far from the control jet; further downstream similar mixing is obtained. Overall, the pulsed jet leads to a similar mixing as the continuous jet. The main positive aspect of pulsed jet is that it requires a reduced mass flow. This example illustrates the application of LES to the study of flow control.

Further developments are in progress to enhance LES application including the use of wall functions and implicit schemes. The value of VLES or DES models will also be assessed.

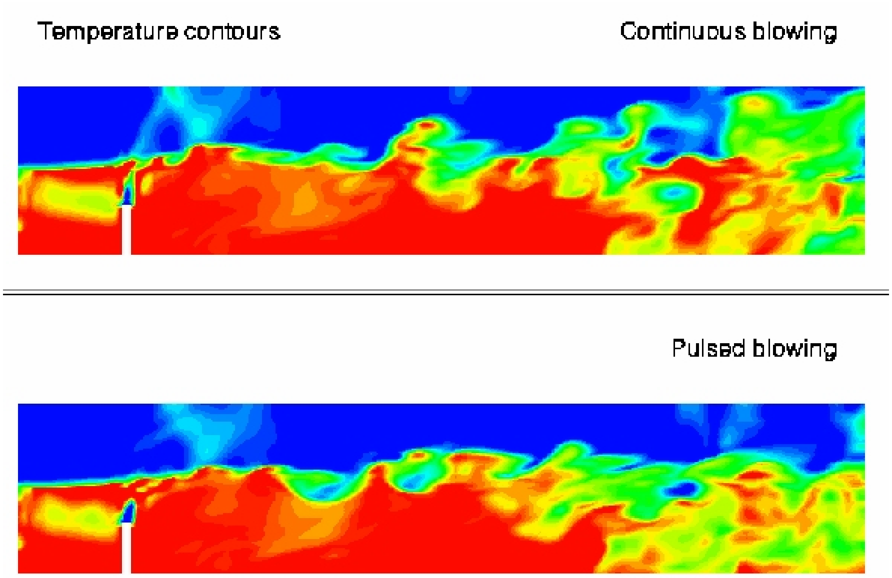

Figure 8 LES simulation of plane shear layer with excitation - Temperature isolines in a plane perpendicular to the mixing layer.

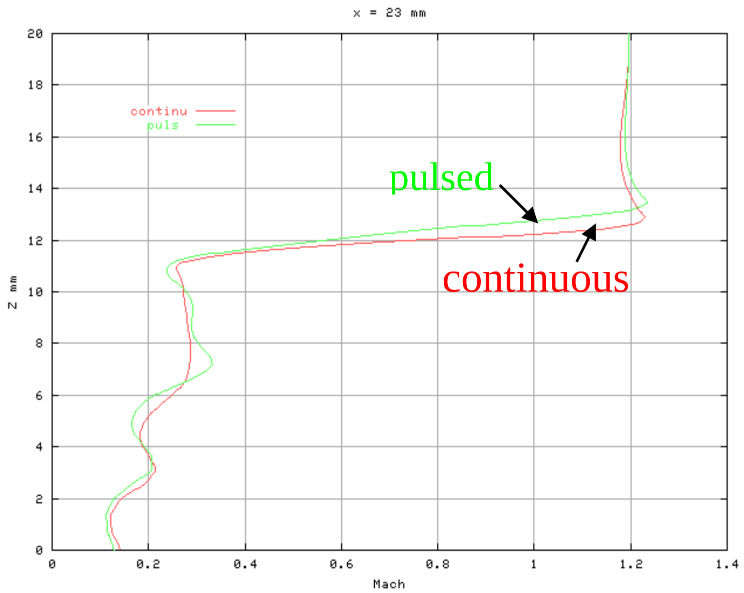

Figure 9 LES simulation of plane shear layer with excitation - Average Mach number profiles 23 mm downstream of the injection jet

5 Conclusions

Recent key advances in simulation have been illustrated via 3 examples: the use of accurate aerodynamics simulation for transonic flutter analysis, the capability of efficient aerodynamic shape optimization for low drag transonic wing design, and the application of accurate LES turbulent modeling for flow control studies.

Multiphysics simulation is already performed in industry as demonstrated by our 1st example. Detailed buffet analysis could have been another example. Progress towards multiphysics optimization is made possible by advances described in our 2nd example.

Acoustics is an important area where multiphysics analysis is needed. Significant work is in progress in this field, part of it brings together CFD and CAA (see [12] for elements on aeroacoustics).

The design of advanced projects like a supersonic business jet will require a high level multiphysics design capability to achieve a quiet and efficient shape. This represents a real challenge in particular to the numerical simulation technology community.

References

[1] F. Chalot, M. Mallet and M. Ravachol, "A Comprehensive Finite Element Navier - Stokes Solver for Low and High-Speed Aircraft Design". AIAA 94-0814.

[2] F. Chalot, B. Marquez,.M. Ravachol, F. Ducros, F. Nicoud, Th. Poinsot, "A consistent Finite Element Approach to Large Eddy Simulation", 29th Fluid Dynamics conference, AIAA 98-2652.

[3] F. Chalot, Q.V. Dinh, M. Mallet, M. Ravachol, G. Rogé, Ph. Rostand, B. Stoufflet, "CFD for aircraft design: recent developments and applications", 4th ECCOMAS Computational Fluid Dynamics Conference, Wiley (1998).

[4] F. Chalot, B. Marquez,.M. Ravachol, F. Ducros, T. Poinsot, "Large Eddy Simulation of a Compressible Mixing Layer : Study of the mixing enhancement", 14th CFD conference, AIAA 99-3358.

[5] E. David, "Modélisation de écoulements compressibles et hypersoniques : une approche instationnaire", Ph.D thesis, INPG, 1994.

[6] Q.V.Dinh, G. Rogé, C. Sevin and B. Stoufflet, "Shape optimization in Computational Fluid Dynamics", European Journal of Finite Elements, Vol 5, 1996, pp 569-594.

[7] Th. Fanion, Ph. D. Thesis, "Etude de la simulation numérique des phénomènes d'aéroélasticité dynamique. Application au problème du flottement des avions". University of Paris Dauphine, 2001.

[8] C. Farhat and M. Lesoinne, "On the accuracy, stability and performance of the solution of three-dimensional nonlinear transcient aeroelastic problems by partitioned procedures". AIAA 96-1388.

[9] "Grand Challenges: high performance computing and communications. The FY 1992 US research and development program". Report by the committee on Sciences. Office of Science and Technology Policy.

[10] T.J.R. Hughes et al., "A New Finite Element Formulation For Computational Fluid Dynamics", Comp. Meth. in Applied Mech. and Eng., North-Holland, 1986.

[11] A. Jameson, "Aerodynamic Design via Control Theory", Journal of Scientific Computing, 3:233-260, 1988.

[12] S. Lemaire and N. Heron, "An overview of Aeroacoustics at Dassault Aviation", Proceeding of the Swing workshop. DLR.

[13] B. Mantel, J. Periaux, M. Sefrioui, B. Stoufflet, J.A. Desideri, S. Lanteri and N. Marco, "Evolutionary Computational Methods for Complex Design in Aerodynamics". AIAA 98-0222.

[14] O. Pironneau, "Optimal Shape Design for Elliptic Systems", Springer - Verlag, New-York, 1984.

[15] M. Ravachol, "Turbulence modeling". Computational Science for the 21th century, Wiley, 1997, pp. 380-391.