INGEGRAF

XVI CONGRESO INTERNACIONAL
DE INGENIERÍA GRÁFICA

AN ITEGRATED METHODOLOGY TO IMPROVE THE DESIGN OF WINGS

E. Pezzuti, A. Ubertini, P.P. Valentini, L. Vita

University of Tor Vergata
Department of Mechanical Engineering
Via del Politecnico, 1
00133 – Roma, Italy
e.mail: vita@ing.uniroma2.it

RESUMEN

El diseño de un alerón de un automóvil para competición deportiva necessita de una metodología capaz tener en cuenta, al mismo tiempo, muchos aspectos, porqué muchas causas contribuen a la prestación global. En este trabajo se ha propuesto un método que integra diferentes criterios de cálculo, CAD-FEM-CFD, para estudiar la aereodinámica de un alerón bajo deformación. Aunque CFD es muy importante por el diseño, muchas veces lo cuerpos, impactados por los fluidos, estan considerados rigidos; pero las deformaciones estructurales pueden cambiar la forma nominal del alerón modificando la fluido-dinámica del sistema. Para las automoviles para competición esto efecto es bienvenido cuando, aumentando la velocidad del vehículo, el ángulo de ataque, las recistencia aerodinámica y la disminución de la sustentación se reducen. En fórmula uno esto efecto esta prohibido y la deformación del alerón, bajo la carga aerodinámica, tiene que estar contenuda bajo un límite establecido. El metodo propuesto emplea el cálculo CFD, actualizando la geometría por los resultados del análisis FEM. El algoritmo desarrollado, automáticamente, aplica el campo de presión, derivado por la solución de la fluido-dinámica, al programa FEM para obtener la deformación de la estructura automáticamente. Esta deformación es utilizada para actualizar la geometría de perfil del alerón y la mesh CFD. El cálculo iterativo termina cuando las deformaciones de dos step consecutivos son más pequeñas que un valor establecido. Normalmente tres o cuatro iteraciones son bastantes. El procedimiento ha sido empleado tambien para evaluar lor effectos de los errores del montaje. La metodología, aplicada a un caso real, ha producido resultados muy interesantes.

Palabras clave: Optimización de la aerodinámica de un alerón, CAD, FEM

ABSTRACT

The design of high performance wing, especially those for car races, requires a methodology able to take into account many aspects at the same time. In many practical cases these aspects

have to be modelled together in order to appreciate the way many causes interact and affect the global performance. In this paper the authors explain a methodology concerning an integrated CAD-FEM-CFD approach to study wing aerodynamics under deformations. Although CFD plays an important role in the design of shape of wing, in many cases the bodies impacted by the fluids are assumed to be rigid; structural deformations can often change the nominal shape of the wing and modify the fluid-dynamics performances of the system. In race car competitions this behaviour is welcome if with the increasing of vehicle speed the angle of attack decreases and both the drag and downforce decrease too. In many regulations this effect is forbidden and wing deformation under aerodynamics load has to be kept under an established limit. The proposed methodology couples CFD analyses with a discrete upgrading of geometry according with the FEM results. The implemented algorithm automatically transfers the pressure field coming from fluid-dynamics solution to FEM solver to obtain the deformation of the entire structure. This deformation is used to update the wing profile geometry and the CFD mesh all over the control volume. Data exchanging and iteration process ends when the deformations of two consecutive steps are smaller than a given error value. The practice shows that it happens quite quickly within three or four iterations. The parameterization of mesh updating has been also tested for studying how assembly errors affect aerodynamics performance. The methodology has been successfully tested on a high performance car front wing and interesting results have been reached.

Keywords: Integrated design, FEM, CFD, wing design

1. Introduction

The efficiency of the aerodynamic parts is a very important issue to obtain good performances from a racing car (DOMINY, 1984; WOLF-HEINRICH, 1993; SHAW, 1988). The efforts of the aerodynamic designers are mainly concentrated to study two forces:

- Downforce (negative lift), which is useful to force the car to the ground, and then to maintain high speed along the curves;
- Drag, which is basically unwanted, because it slows the car.

The capability of an aerodynamic engineer is strictly related to the achievement of a good compromise between lift and drag, or, in other words, to obtain the desired lift with the lowest drag as possible.

The efficiency of a wing in general depends on the following items:

- Aspect ratio (length-width ratio). The amount of downforce produced by a wing is determined by its size. The larger the wing is the greater the downforce is. The higher the aspect ratio is the more efficient the wing is because of less air resistance created by the vortex at the wing tips;
- Angle of attack. The greater the angle of attack is, the greater the downforce is until stall, the greater the unwanted drag is;
- Drag. Increasing downforce, a wing also increases unwanted drag. Drag increases with the angle of attack.

The design of the front wing of a racing car is a critical task since it is the very first part of the car which meets the flow. Moreover its shape affects also the performances of the whole car body and in particular way those of the rear wing. For these reasons the design of a front wing costs about twice more than the design of the rear one.

Each front airfoil is made of one or more mainplanes of carbon fibres almost covering the entire width of the car, and generally has one full spanning flap; endplates constitute the boundaries of each mainplane. The performance of the front-wing changes radically with the closeness to the ground for the well-known "ground effect"; the distance between the wing and the ground is usually limited by specific competition regulations. Developments of the wing design usually concentrate on the wing profile and the flaps; moreover, some teams recently angled the leading edge to form a forward facing V-shape, in order to exploit the full potential of the wing: flow visualizations in fact showed that the suction power of the wing is so strong that air approaches to the leading edge with a different angle from the one presented by the undisturbed flow. Endplates design tends to change a lot: their primary function consists in separating the high pressure air (top of the wing) from the low pressure air (bottom of the wing); some teams also use "splitters" attached to the undersurface of the wing to assist the endplates. CFD then would be an ideal tool for design, and it is already used and growing in importance because it allows to reduce time and cost within such a complex scenario. In fact, the performance of a number of possible configurations of the wing can be evaluated with the aid of a computer architecture; then only efficient configurations undergo wind-tunnel tests after which the optimal solution can be chosen. In many cases CFD is not sufficient to simulate the behaviour of the wing impacted by an air flow. In fact, the elasticity of the wing may cause deformations (ANDREASSI et al., 2004) which affect the geometrical shape and position of the wing and modify the aerodynamic properties. In this case an integrated approach (GUBERT et al., 2001, PAHL et al., 1999) is the only way to face the problem in a correct manner. On the other hand a numerical code which solves both fluid dynamics and structural models at the same time requires a lot of hardware resources and it is difficult to be optimized for both problems at the same time too.
For this reason the development of an integrated design methodology which couples both aspects, but solving each problem using a independent solver, seems to be a winning strategy.

2. Integrated design methodology

We can summarize the proposed methodology in seven steps.

1. CAD modelling: the shape of the wing is drawn using a CAD program or its geometry is acquired by accurate measurements and then reproduced in a CAD program. Then, the elements for structural and fluid dynamic analyses are generated. It is important to notice that the mesh is generated only at this step, no other meshes have to be computed. So the computational effort is reduced.
2. A preliminary Normal Modes of Vibration analysis (MASTRACCO, 2003) is performed in order to avoid the possibility of dynamic amplification of structural deformations due to flutter or similar phenomena.
3. The wing is then analysed by means of the 3D CFD code to define the pressure field around the structure;
4. The pressure field is converted into nodal loads to be applied to the finite element model;

5. A linear elastic analysis is performed in order to find the deformation of the entire wing under the nodal loads defined at step 4;
6. The deformations evaluated at step 5 are processed by an algorithm program (DE SIMONE, 2003) in order to update the entire fluid dynamics volume mesh. The mesh is only updated. The topological information and the element appearance are the same.
7. CFD analysis is again performed to define the updated pressure field around the deformed structure.

Steps from 3 to 7 are repeated until the convergence, i.e. the same deformation of the previous step within a predefined tolerance, is obtained.

Cad Modelling

The geometry of the wing has been splitted into several surfaces which have been defined using their bounding curves (non uniform B-splines) and a cubic interpolation. The entities have been exported in a general purpose ACIS format (CORNEY et al., 2001) in order to preserve their topological information about the hierarchy and orientation. The used CAD model of wing is depicted in Figure 1; it shows a "spoon" shape, i.e. it is slightly lower next to the symmetry axis. According to Formula 1 regulations, and with the additional hypothesis that the car flat bottom lies at 15.20 mm from the track, the distance between the lowest portion of the wing and the track was assumed to be equal to 11.7 cm.

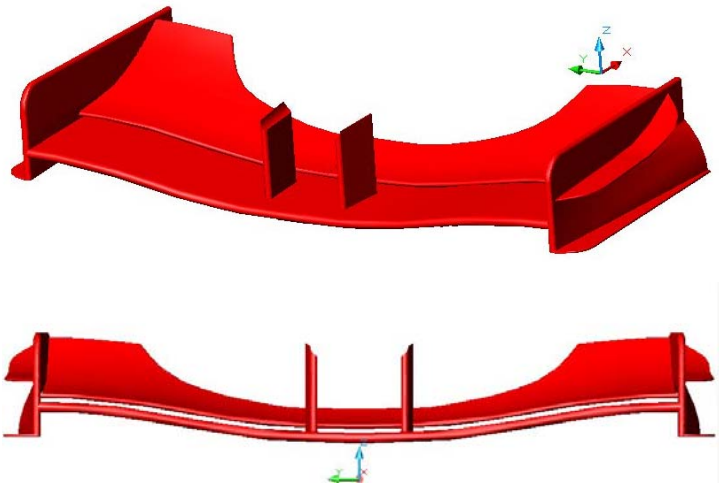

Figure 1. CAD model of the entire front wing (3D view and front view)

FEM Model

In order to study the deformation under pressure loads, the front wing has been modelled by means of finite element technique. The system has been simulated using triangular planar element with 5 d.o.f. each node. Only an half of the entire wing has been meshed, because the symmetry of the geometry, the loads and the constrains simplify the analysis. The model has been constrained in the cutting plane to preserve the above mentioned conditions. Moreover, the node of the vertical anchor segment to the nose has been fixed to the ground. The material used is a generic CFRP with no

oriented fibres, so that it could be modelled as an isotropic linear elastic material, characterised by the following properties:

- Young's modulus (E): 2.28E+11 Pa
- Shear modulus (G): 1.03E+10 Pa
- Poisson ratio (ν): 0.27
- Mass density (ρ): 1580 kg/m^3

An overall picture of the undeformed mesh is presented in Figure 2.
The preliminary vibration analysis shows that the first normal frequencies do not have resonance phenomena due to flutters or vortex sheddings for car speed up to 400 km/h. A complete aero-elasticity analysis can be found in MASTRACCO, 2003. The first two normal modes deformation is depicted in Figure 3.

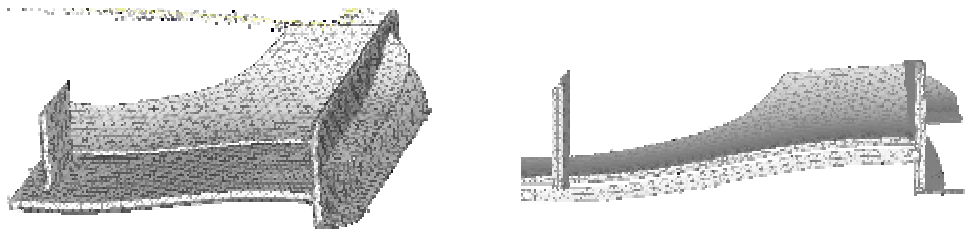

Figure 2: Undeformed mesh of the half wing

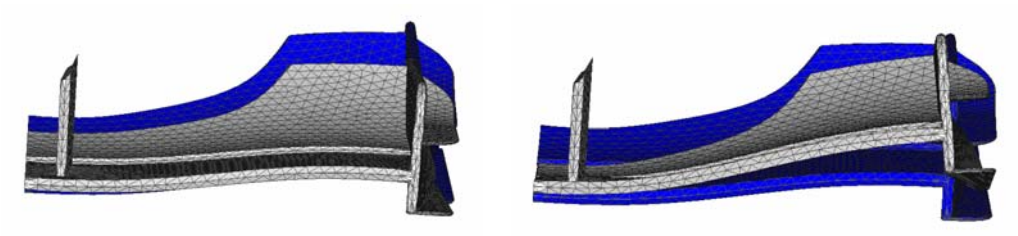

Figure 3: First vibrational modes of the wing (First at 39.8 Hz, on the left and second at 78.4 Hz on the right).

CFD Model

Since the speed of the vehicle is quite high, it surely induces turbulent phenomena around the wing: apart from standard conservation relations, additional equations are required to simulate these not negligible effects. The chosen turbulent model was the RNG $k-\varepsilon$ model (RAMNEFORS, 1996) because it represents a compromise between accuracy and time-cost. It is linked to the fluid dynamics equations by means of the classical turbulent viscosity hypothesis which is:

$$\nu_t = C_\mu \frac{k^2}{\varepsilon}$$

where for this model $C_\mu = 0.0845$. Inlet boundary conditions on turbulent quantities are defined on the basis of a percentage of the average kinetic energy $T_{in} = 0.5 u_i u_i$ and of a characteristic length L by means of the following hypothesis:

$$k_{in} = f \times T_{in} \qquad \varepsilon_{in} = \frac{C_{\mu}^{3/4} \times k_{in}^{3/2}}{0.07L}$$

where f assumes values in the range 0.01 ÷ 0.1, depending on the inlet turbulence level. The fluid dynamic mesh is made of tetrahedral elements: this model in fact allows to obtain a quite refined grid near the wing surface which surely constitutes the portion of the domain most interested by high variable gradients. The element faces on the surface of the wing are coincident with the triangular elements used in the FEM approach to obtain a congruency among boundary nodes. Since turbulence plays a dominant role in the transport of mean momentum and other scalars, it has been necessary to be sure that turbulent quantities are properly resolved. For this reason a preliminary analysis has been performed and results have been compared to those coming from experiments.

Connecting algorithm

The algorithm which allows to transfer data between two solvers and also to perform the mesh updating is a subroutine written in FORTRAN language. Data translation between the FEM and CFD solver is quite simple because both programs work with ASCII input files. So it is possible to read and process information in an easy way, just keeping format rules and keywords to define objects. Automatic mesh updating is a more complex task. The proposed idea is the following:

1. We start from the basic CFD mesh, which is defined by means of node and elements list and topological information about connectivity among elements.
2. Making the hypothesis that deformation of the wing are quite small, we can assume that the topological connections are the same in whatever conditions. So this information does not need to be updated.
3. Reading the deformations coming from structural solver we can perform this transformation: the nodes on the wing geometry move according to the computed deformation; the nodes on the control volume boundary do not move; nodes in the space between wing and outer boundary move according with proximity rules. It consists to move a node with a displacement law which is the linear combination of the displacements of the nodes on the wing surface weighted with proximity multipliers. It means that the more a node is near to the airfoil surface the more it moves, and it moves with a function depending on the displacement of the nodes in the neighbourhood (DE SIMONE, 2003).
4. Special checks have been introduced to avoid negative volume elements generation or excessive deformation of cells.

Rigid wing fluid dynamic analysis

Pure fluid dynamics results are here displayed for the rigid geometry: the analysis was performed for speeds up to 300 km/h. Lift and drag are reported in Figure 5, while some sketches concerning pressure fields are reported in Figure 4.

Deformed wing FEM and CFD analysis

The pressure load coming from preliminary CFD analyses has been transferred as nodal load to the explained FEM model. These data were used to perform a stress-strain analysis of the wing. The deformation of each nodes has been acquired by the connecting algorithm which has been performed an updating of the fluid-dynamic mesh according to the rules previously discussed. So it was possible to run another CFD analysis with a slightly different geometry. The pressure field is then compared to those obtained in the previous analysis. In case of accordance between the two results the iteration stops and the last results are assumed to be the right results. If there is an unacceptable difference the new pressure field is transferred to FEM solver and another structural analysis is performed. The practice shows that the iteration process takes three or four steps to be completed.

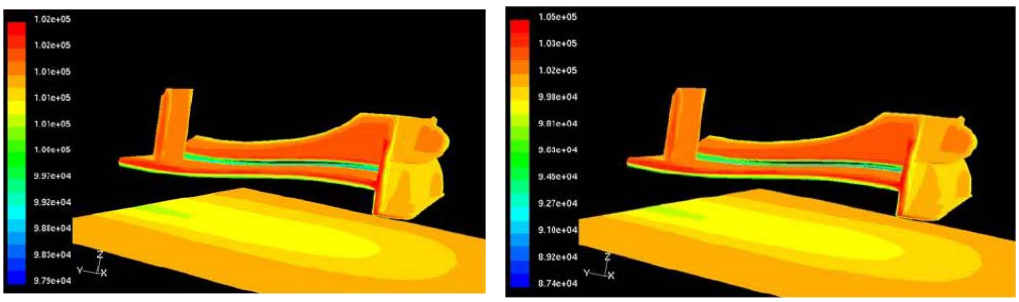

Figure 4: Static pressure fields around the wing: at 100 km/h on the left, at 200 km/h on the left); values are in Pascal

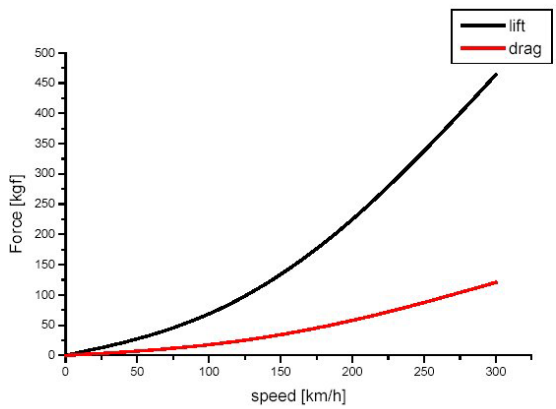

Figure 5: Lift and drag of the wing at different speed

In Figure 6, lift and drag of the deformed wing as function of speed are presented. The deviation from the rigid wing curves gets wider by raising the speed. As it can be observed, the predicted fluid dynamic forces offered by the wing change in a not negligible way by using an integrated approach: the deviation is obviously wider at higher speeds; it can be observed that at 300 km/h drag reduces by 5.5%, while lift reduces by 3.5%: the deformed wing then presents a higher efficiency than the rigid,

even if it produces less desired effect, i.e. the lift. This circumstance would even justify the definition of an inverse numerical procedure which could give the undeformed wing geometry starting from the desired lift and the deformed configuration.

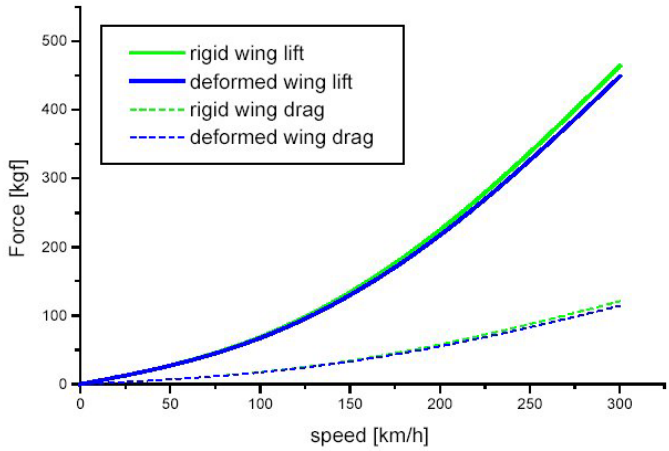

Figure 6: Comparison between lift and drag of the rigid and deformable wing at different speed

The FEM results show that the critical areas are the lower airfoil and the nose connection zone where elements show higher stress values. In this case the deformation of the wing (shown in Figure 7) decreases the attack angle and explains the changing of drag and downforce values.

3. Assembly errors analysis

The connecting algorithm allows also to perform analyses concerning the influence of assembly errors on wing performances (both drag and downforce). For this purpose starting from the same basic mesh the algorithm can update the position of the nodes simulating the presence of an angular misalignment. This one causes a variation of angle of attack of the airfoils. Also for these analyses the wing is assumed to be deformable and the effect of deformation is taken into account with the same methodology. Figure 7 shows the graphs about the results of the analyses, and Tables 1 and 2 summarize numerical results of the same analyses.

4. Conclusions

A numerical fluid dynamic and structural coupled approach to study wings (or in general, objects in a fluid flow) capable of being deformed has been presented in this paper. The final result consists of an identification of the actual wing geometry and a more realistic flow field around the object The presented integrated methodology results mainly indicate that:

- The use of the integrated methodology is particularly suggested for high velocity vehicles (i.e. 300 km/h), while little difference in the fluid dynamic fields is observed at ordinary speeds (i.e. 100 km/h).

- At high speeds lift and drag parameters are sensibly influenced by the wing deformation, since differences in the range of 4% - 5% have been observed by applying the integrated methodology with respect to fluid dynamic pressure loads.
- The low number of iterations required to achieve the steady fluid dynamic and deformation fields makes the use of the integrated procedure cost-effective especially in terms of time. Moreover this procedure allows the reduction of the number of possible configurations to be tested in the wind-tunnels, which remain the decisive test to be get through.

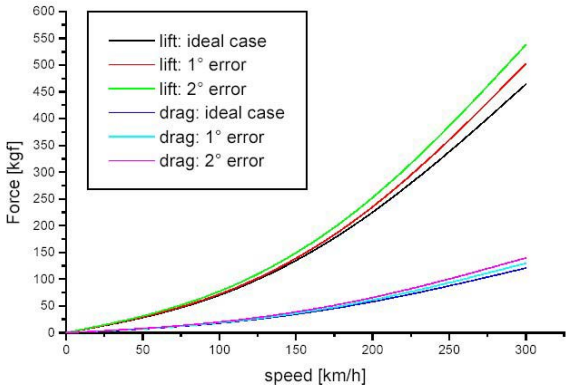

Figure 7: Lift and drag for deformable wing with mechanical errors of inclination

Table 1: Lift of deformable wing as function of speed and assembly error (values are in kgf).

	100 km/h	**200 km/h**	**300 km/h**
No error	50.77	200.89	448.58
+1° error	54.47	205.90	480.88
+2° error	58.38	220.80	512.20
-1° error	45.87	117.20	412.92
-2° error	42.22	160.98	375.82

Table 2: Downforce of deformable wing as function of speed and assembly error (values are in kgf).

	100 km/h	**200 km/h**	**300 km/h**
No error	13.00	51.10	114.20
+1° error	13.95	53.52	122.59
+2° error	15.00	57.55	130.65
-1° error	12.12	47.15	108.12
-2° error	11.44	44.19	101.13

Results obtained are encouraging and suggest to further investigate the possibilities offered by this procedure. For example the definition of an inverse procedure which could provide the undeformed geometry starting from the deformed one and the given constraints in terms of lift required.

References

ANDREASSI, L., MULONE, V., VALENTINI, P.P., VITA, L., (2004) *A CFD-FEM Approach to Study Wing Aerodynamics under Deformation*, Proceedings of SAE 2004-01-0444

CORNEY, J., LIM, T., (2001) *3D Modeling with ACIS*, Saxe-Coburg Pubblications.

DE SIMONE, G. (2003) *Analisi Integrata dell'interazione Fluido-Struttura di un Alettone per Autovetture ad Elevate Prestazioni* (in Italian). First level degree in Mechanical Engineering, University of Rome "Tor Vergata", a.y. 2002-2003

DOMINY, R.G., (1984) *Aerodynamics of Grand Prix Cars*, Proc. Instn. Mech. Engrs.1984 198D 1-7

GUMBERT, C.R., HOU, G.J.W., NEWMAN, P.A., (2001) *Simultaneous Aerodynamic Analysis and Design Optimization (SAADO) for a 3-D Flexible Wing*, 39th Aerospace Science Meeting & Exhibit, Paper n. AIAA 2001-1107.

MASTRACCO, A. (2003) *Analisi del comportamento vibrazionale e fenomeni aereoelastici di un alettone per autovetture da competizione* (in Italian). First level degree in Mechanical Engineering, University of Rome "Tor Vergata", a.y. 2002-2003

PAHL, G., BEITZ, W., (1999) *Engineering Design, a Systematic Approach*, Springer ed.

RAMNEFORS, M. et al. (1996), *Accuracy of Drag Predictions on Cars Using CFD - Effect of Grid Refinement and Turbulence Models*, SAE Paper 960681

SHAW, C.T., (1988) *Predicting Vehicle Aerodynamics Using Computational Fluid Dynamics – A User's perspective*. Automotive Aerodynamics - Society of Automotive Engineers

WOLF-HEINRICH, H., (1993) *Aerodynamics of Road Vehicles*, Fluid Mechanics Annual Review 25: 485-537