Proceedings of the XI DINAME, 28th February-4th March, 2005 - Ouro Preto - MG - Brazil
Edited by D.A. Rade and V.Steffen Jr

AN AEROELASTIC ANALYSIS METHODOLOGY FOR TWO-DIMENSIONAL LIFTING SURFACES

Alexandre Noll Marques
Instituto Tecnológico de Aeronáutica, CTA/ITA
12228-900 - São José dos Campos - SP - Brazil
noll@redecasd.ita.br

Carlos Frederico Castro Simões
EMBRAER - Empresa Brasileira de Aeronáutica, Av. Brigadeiro Faria Lima, 2170
12227-901 - São José dos Campos - SP - Brazil
fred.simoes@pobox.com

João Luiz F. Azevedo
Instituto de Aeronáutica e Espaço, CTA/IAE/ASE-N
12228-904 - São José dos Campos - SP - Brazil
azevedo@iae.cta.br

Abstract. *This work is a continuation of an effort of developing a methodology for the aeroelastic analysis of two-dimensional lifiting surfaces using an unsteady, Euler-based, CFD tool for the calculation of the aerodynamic operator. The CFD tool solves the flow problem with the finite-volume method applied in an unstructured grid context. Details on space discretization and time stepping scheme are discussed and references to further details on artificial dissipation modeling and convergence acceleration techniques, available for the steady-case situations, are cited. The computational mesh is obtained with the commercial generation tool ICEM CFD©. The proposed methodology is based on the determination of the aerodynamic operator with the transfer function technique, which is given, in the frequency domain, by the analysis of the system's response to an exponentially-shaped pulse in the time domain. The response in the frequency domain is achieved with the Fast Fourier Transform (FFT) technique available in any mathematical manipulation tool, such as Matlab©. Some numerical experiments are performed involving unsteady transonic and subsonic flows around a NACA 0012 airfoil and the results are presented as curves of aerodynamic generalized forces. The unsteady simulations start from a converged steady state solution obtained by the same CFD tool. These unsteady results are compared with available data in the literature. The respective Fourier transforms are also determined.*

Keywords: *Aeroelasticity, Unstructured Meshes, Finite Volume Technique, CFD*

1 Introduction

Aeroelasticity can be defined as the science which studies the mutual interaction between aerodynamic and dynamic forces. However, a broader definition is possible when inertial forces are also focused. The analysis of dynamics characteristics of either complex or simple structures are quite developed nowadays as far as numerical and experimental methods are concerned. Hence, it is correct to state that reliability in aeroelastic calculations is strongly dependent to the correct evaluation of the aerodynamic operator.

Traditionally, the methods developed for determining the aerodynamic operator for subsonic and supersonic regimes are based on linearized formulations which do not present the same satisfactory results in the transonic range. According to Tijdeman (1977), this occurs due to the nonlinearity of transonic flows caracterizing a significative alteration of the flow behavior, even when a profile is submited to small pertubations. Thus being, the methodology here presented, which is based on the ideas of Raush, Batina and Yang (1990), intends to obtain the aerodynamic operator for two- dimensional lifting surfaces employing modern CFD techniques.

Computational Fluid Dynamics (CFD) is a subject that has played an extremely important role in recent aerodynamics studies. The possibility of treating numerically a broad range of phenomena which occur in flows over bodies of practically any geometry has inumerous advantages over experimental determinations, such as greater flexibility together with time and financial resources savings.

However, obtaining more reliable numerical results for a growing number of situations has been one of the major recent challenges in many science fields. Fletcher (1988a) and Hirsch (1994) show that particulary in aerodynamics, the general phenomena are governed by the Navier-Stokes equations, which constitute a system of coupled nonlinear partial differential equations that has no general analytical solution and that is of difficult algebraic manipulation. Hirsch (1994) coments, among other issues concerning CFD techniques, on how to simplify the mathematical models conveniently in order to ease the numerical treatment of each case. A survey on this subject is also noted by Azevedo (1990). Space and time discretization schemes, as well as convergence acceleration techniques, boundary condition estabilishment and other numerical integration tools are available and largely used in order to solve such models.

After selection of the theoretical model, it is indispensable to define the physical domain where the flows take place, determining the boundary conditions. This solution approach demands the discrete representation of the physical domain to make the problems numerically coherent defining a computational mesh of points or regions where the calculations are performed. The mesh generation, as it is vastly documented in the literature, *i.g.* Fletcher (1988b), and verified by the CTA/IAE work group's own experience, is extremely important and decisive in the accuracy and convergence of the solution. The mesh kind is also an essential factor on the CFD tool behavior. Structured meshes have the advantage of being well-behaved, the existence of an intrinsic correspondence between adjacent nodes and a very good control over grid refinement through stretching functions. They are widely employed together with finite difference codes due to its nature when neighbor nodes relations are concerned, although there are no restrictions on their use with other methods. However, this sort of mesh do not adpat readily to complicated geometries, requiring the adoption of sophisticated multiblock mesh techniques. On the other hand, unstructures meshes are extremely flexible when it comes to geometric forms and they allow the use of interesting techniques such as adaptative refinement. The way the neighbor cells and conectivity data is arranged in this sort of mesh leads naturally to its use together with finite volume methods. However, as in the structured case, there are no restrictions to its employment with other formulations.

As stated by Marques (2004), together with the evolution of the work and projects performed by CTA/IAE, the demand for aerodynamic parameters has swelled, mainly those concerning the vehicles developed in this center. Nevertheless, the application of CFD tools in these parameter analyses has always been limited by the need of adequated code development and the lack of computational resources compatible with the work to be performed. Therefore, a progressive approach has been adopted in the development of CFD tools in CTA/IAE and in ITA, as presented by Azevedo (1990), Azevedo, Fico and Ortega(1995), Fico and Azevdo (1994), Azevedo, Strauss and Ferrari (1997), Bigarelli, Mello and Azevdo (1999), Bigarelli and Azevedo (2002), Oliveira (1993), Simões and Azevedo (1999).

The present work is based on the finite volume formulation, where a CFD tool is applied with unstructured bidimensional meshes around lifting surfaces to acquire nonstationary responses to harmonic and impulsed motions. These time-domain responses supply the generalized aerodynamic forces necessary as input to the aeroelastic model. The methodology here presented intends to obtain frequency-domain responses from the solutions to impulsed motion and, with that information, determine the aeroelastic stability margin with a single expensive CFD run.

2 Aerodynamic Theoretical Formulation

The CFD tool applied in this work is based on the Euler equations for the two-dimensional case. Due to the use of unstructured meshes and the adoption of the finite volume approach, these equations were used in the Cartesian form. Besides, as usual in CFD applications, flux vectors are employed and the equations are nondimensionalized. Hence, they can be written as

$$\frac{\partial}{\partial t}\int\int_V Q dx dy + \int_S (E dy + F dx) = 0. \tag{1}$$

In Eq. (1), V represents the volume of the control volume or, more precisely, its area in the two-imensional case. S is its surface, or its side edges in 2-D. Q is the vector of conserved properties, given by

$$Q = \left\{ \ \rho \ \rho u \ \rho v \ e \ \right\}^T . \tag{2}$$

E and F are the flux vectors in the x and y directions, respectively, defined as

$$E = \left\{ \begin{array}{c} \rho U \\ \rho u U + p \\ \rho v U \\ (e+p)U + x_t p \end{array} \right\}, \quad F = \left\{ \begin{array}{c} \rho V \\ \rho u V \\ \rho v V + p \\ (e+p)V + y_t p \end{array} \right\}. \tag{3}$$

The nomenclature adopted here is the usual in CFD: ρ is the density, u and v are the Cartesian velocity components and e is the total energy per unity of volume. The pressure (p) is given by the perfect gas equation, written as

$$p = (\gamma - 1)\left[e - \frac{1}{2}\rho\left(u^2 + v^2\right)\right]. \tag{4}$$

Once again, as usual, γ represents the ratio of specific heats. The contravariant velocity components (U and V) are determined by

$$U = u - x_t \qquad \text{an} \qquad V = v - y_t, \tag{5}$$

where x_t and y_t are the Cartesian components of the mesh velocity in the nonstationary case. For further details on the theoretical formulation, such as boundary and initial conditions, please refer to Marques (2004).

3 Numerical Formulation

The algorithm presented here is based on a cell-centered finite volume scheme in which the stored information is actually the variable avarage value throughout the entire control volume. These mean values are defined as

$$Q_i = \frac{1}{V_i} \int\int_{V_i} Q dx dy. \tag{6}$$

Equation (1) can, then, be rewritten for each $i-th$ cell as

$$\frac{\partial}{\partial t}\left(V_i Q_i\right) + \int_{S_i} \left(E dy - F dx\right) = 0. \tag{7}$$

The remaining integration in Eq. (7) represents the flux of the vector quantities E and F through each control volume's boundary. This code was developed to be used with unstructured meshes composed of triangles. The flux, however, can be evaluated as the sum of the flux contributions of each edge, which is obtained approximately from the average with the neighbors conserved quantities, as proposed by Jameson and Mavriplis (1986). Hence, a convective operator is defined and it is given by

$$\int_{S_i} \left(E dy - F dx\right) \approx C\left(Q_i\right) = \tag{8}$$

$$\sum_{k=1}^{3} \left[E\left(Q_{ik}\right)\left(y_{k_2} - y_{k_1}\right) - F\left(Q_{ik}\right)\left(x_{k_2} - x_{k_1}\right)\right], \tag{9}$$

where

$$Q_{ik} = \frac{1}{2}\left(Q_i + Q_k\right), \tag{10}$$

and the (x_{k_1}, y_{k_1}) and (x_{k_2}, y_{k_2}) coordinates are relative to the vertices which define the interface between the cells.

The Euler equations are a set of nondissipative hyperbolic conservation laws. Hence, as given by Pulliam (1986), their numerical treatment requires an inherently dissipative discretization scheme or the introduction of artificial dissipation terms in order to avoid oscillations near shock waves and to damp high frequency uncoupled error modes. Azevedo and Oliveira (1993) state that the flux evaluation method adopted in the present CFD tool is analogous to a centered difference scheme in finite difference formulation. In this case, Pulliam (1986) shows that there is the necessity of adding artifitial dissipacion terms. Details on the adopted artificial dissipation scheme are given in Marques (2004).

The numerical solution is advanced in time using a second-order accurate, 5-stage, explicit, hybrid scheme which evolved from the consideration of Runge-Kutta time stepping schemes (Jameson, Shmidt and Turkel, 1981, and Mavriplis, 1988). This scheme, already including the necessary terms to account for changes in cell area due to mesh motion or deformation (Batina, 1989, and Batina, 1991), can be written as

$$Q_i^{(0)} = Q_i^n \tag{11}$$

$$Q_i^{(1)} = \frac{V_i^n}{V_i^{n+1}} Q_i^{(0)} - \alpha_1 \frac{\Delta t_i}{V_i^{n+1}} \left[C\left(Q_i^{(0)}\right) - D\left(Q_i^{(0)}\right)\right] \tag{12}$$

$$Q_i^{(2)} = \frac{V_i^n}{V_i^{n+1}} Q_i^{(0)} - \alpha_2 \frac{\Delta t_i}{V_i^{n+1}} \left[C\left(Q_i^{(1)}\right) - D\left(Q_i^{(1)}\right)\right] \tag{13}$$

$$Q_i^{(3)} = \frac{V_i^n}{V_i^{n+1}} Q_i^{(0)} - \alpha_3 \frac{\Delta t_i}{V_i^{n+1}} \left[C\left(Q_i^{(2)}\right) - D\left(Q_i^{(1)}\right)\right] \tag{14}$$

$$Q_i^{(4)} = \frac{V_i^n}{V_i^{n+1}} Q_i^{(0)} - \alpha_4 \frac{\Delta t_i}{V_i^{n+1}} \left[C\left(Q_i^{(3)}\right) - D\left(Q_i^{(1)}\right) \right] \quad (15)$$

$$Q_i^{(5)} = \frac{V_i^n}{V_i^{n+1}} Q_i^{(0)} - \alpha_5 \frac{\Delta t_i}{V_i^{n+1}} \left[C\left(Q_i^{(4)}\right) - D\left(Q_i^{(1)}\right) \right] \quad (16)$$

$$Q_i^{n+1} = Q_i^{(5)}, \quad (17)$$

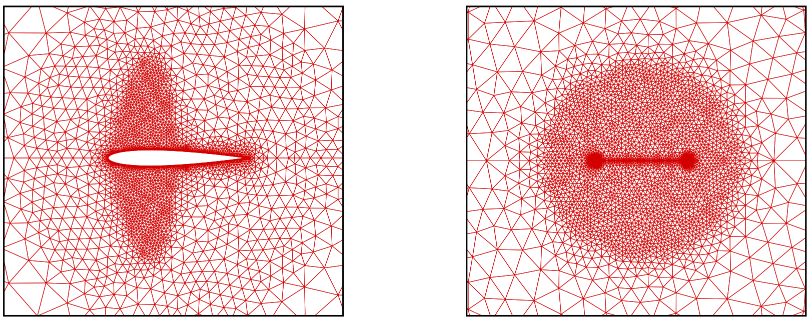
where the superscripts n and $n+1$ indicate that these are property values at the beginning and the end of the $n-th$ time step, respectevily. $D(Q)$ denotes the artificial dissipation operator. The values used for the α coefficients, as suggested by Mavriplis (1990), are

$$\alpha_1 = \tfrac{1}{4}, \alpha_2 = \tfrac{1}{6}, \alpha_3 = \tfrac{3}{8}, \alpha_4 = \tfrac{1}{2}, \alpha_5 = 1. \quad (18)$$

In Eq. (17) the convective operator, $C(Q)$, is evaluated at every stage of the integration process, but the artificial dissipation operator, $D(Q)$, is only evaluated at the two initial stages. This is done with the objective of saving computational time, since the evaluation of the last is rather expensive. As discussed by Jameson, Shmidt and Turkel (1981), this type of procedure is known to provide adequate numerical damping characteristics while achieving the desired reduction in computational cost. Steady state solutions for the mean flight condition of interest must be obtained before the unsteady calculation can be started. Therefore, it is also important to guarantee an acceptable efficiency for the code in steady state mode. In the present work, both local time stepping and implicit residual smoothing (Jameson and Mavriplis, 1986, Jameson and Baker, 1983 and Jameson and Baker, 1987) are employed to accelerate convergence to steady state. More details on convergence acceleration techniques are found in Oliveira (1993) and Marques (2004).

4 Mesh Generation and Movement

The meshes used in the present work were generated with the commercial grid generator ICEM CFD©, a very powerfull tool capable of creating sophisticated meshes with very good refinement and grid quality control. Figures 1(a) and 1(b) show the meshes around a NACA 0012 profile and a flat plate, respectively, which are employed to obtain the results here presented.



(a) (b)

Figure 1: Mesh around (a) NACA0012 profile with 292 wall points and (b) flat plate with 236 wall points.

Unsteady calculations involve body motion and, therefore, the computational mesh should be somehow adjusted to take this motion into account. The approach adopted here is to keep the far field boundary fixed and to move the interior grid points in order to accomodate the prescribed body motion. This was done following the ideas presented by Batina (1989), and Rausch, Batina and Yang (1990), which assume that each side of the triangle is modeled as a spring with stiffness constant proportional to the length of the side. Hence, once points on the body surface have been moved and assuming that the far field boundary is fixed, a set of static equilibrium equations can be solved for the position of the interior nodes. Control volume areas for the new grid can, then, be computed. The mesh velocity components can also be evaluated considering the new and old point positions and the time step.

5 Aeroelastic Analysis Methodology

Due to the lack of space, the detailed aeroelastic formulation is not presented in this work. For such details, please refer to Oliveira (1993) or Bisplinghoff, Ashley and Halfman (1955). Starting with the assumption that the unsteady movements related to the aeroelastic phenomena, mainly flutter, can be approximated by harmonic motions, a large computational cost reduction comes from the use of the indicial method. According to this approach, the aerodynamic response to an harmonic excitation of any frequency can be obtained from Duhamel's integral of the response of the flow to a indicial motion. Following this same idea, and noticing that the transient flow due to an implusive excitation takes a shorter period of time to die out than those that results from an indicial motion, Oliveira (1993) proposed a similar methodology where the aerodynamic response are evaluated in the frequency domain from the response to a implusive excitation in the time domain.

Therefore, the aerodynamic calculations for a determined flight condition are reduced to a single computational run for each structural mode. Moreover, the only linearity hipothesis adopted is of the aerodynamic generalized forces to the displacement modes and amplitudes, which guarantees that this methodology captures the flow's nonlinearities and dynamics, except for those related to viscous effects which are not included in an Euler formulation.

Nevertheless, the theoretical impulse movement is a singularity and the indicial one leads to the appearance of infinite velocities, which makes both numerically untreatable. Hence, other smoother excitation functions are employed (Davis and Salmond, 1980, and Mohr, Batina and Yang, 1989). The motion suggested by Bakhle *et al.* (1991) is defined as

$$f_p(\bar{t}) = \begin{cases} 4\left(\frac{\bar{t}}{\bar{t}_{max}}\right)\exp\left(2 - \frac{1}{1-\frac{t}{\bar{t}_{max}}}\right); & 0 \leq t < t_{max} \\ 0; & t \geq t_{max} \end{cases} \tag{19}$$

where the overline indicates the dimensionless time and $\bar{t}_{max}$ is the impulse duration. As can be seen in Fig. 2, the function defined in Eq. (19) guarantees a smooth movement.

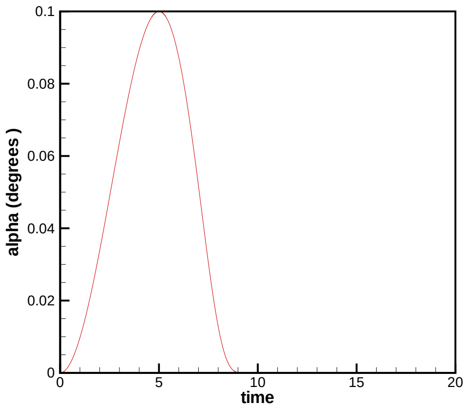


Figure 2: Attack angle impluse excitation.

The methodology defined as analysis through transfer function by a impulse consists, then, of the following steps:

- Obtaining the steady aerodynamic solution for a given Mach number and attack angle;
- Performing unsteady aerodynamic response evaluation departing from the steady solution given in the previous item. This stage leads to time responses in terms of aerodynamic coefficients as result to a impulse excitation for each of the modes;
- Obtaining the Fourier transform of the time responses applying a Fast Fourier Transform (FFT) algorithm. This is done in the present work employing the FFT capability available in the commercial programm Matlab©. As shown by Oliveira (1993), the corresponding frequency domain is given by

$$f = \frac{1}{\Delta t}\frac{i}{N} = \frac{a_\infty}{\Delta \bar{t} c}\frac{i}{N};\, i = 0, 1, 2, ..., N_{max}, \tag{20}$$

$$N_{max} = \begin{cases} N/2; & \text{if } N \text{ even} \\ (N-1)/2; & \text{otherwise} \end{cases} \tag{21}$$

where c is the chord lentgh and a_∞ the freestream speed of sound. Equation (20), rewritten in terms of the reduced frequency based on the chord length, stands as

$$k = \frac{\omega c}{U_\infty}, = \frac{2\pi f c}{U_\infty} \tag{22}$$

$$k = 2\pi M_\infty \Delta\bar{t}\frac{i}{N};\ i = 0, 1, 2, ..., N_{max}, \tag{23}$$

where M_∞ is the Mach number refering to the undisturbed conditions.

As the input is not exactly an implusive excitation, the real impulse response is evaluated using the Duhamel's integral concept and a well-known property of the convolution integral, as given by Brigham (1988)

$$g(t) = fp(t) * i(t) \Rightarrow G(f) = Fp(f)I(f), \tag{24}$$

$$I(f) = \frac{G(f)}{Fp(f)}. \tag{25}$$

where $i(t)$ represents the time response to a implusive movement and $g(t)$ is the response to the impulse excitation given in Eq. (19). The functions in capital letters are Fourier transforms of the corresponding functions in low case letters. Therefore, after obtaining the FFT of the time response, it has to be divided by the FFT of the input impulse function. Although the input is not the exact impulse excitation, it is capable of exciting the reduced frequencies of interest in aeroelastic studies;

- Aproximating the obtained data with an interpolating polynomial, shown in Oliveira (1993);
- Formulation of the corresponding eigenvalue problem, valid for a determined range of dimensionless velocities, and, finally, flutter speed prediction through a root locus analysis.

6 Results and Discussion

Before attempting applications of the proposed methodology, some validation simulations were performed with the CFD tool. This has been done throughout the entire development of this code as can be seen in Azevedo (1992), Oliveira (1993) and Simões and Azevedo (1999). The present authors proceeded with this effort obtaining the results shown in Figs. 3 and 4 for a NACA 0012 profile at $M_\infty = 0.755$ performing a $2.51°$ amplitude harmonic pitching movement about the quarter-chord with $k = 0.1628$. The authors point out that k is the reduced frequency based on the chord length, defined in Eq. (22). Figure 3 presents the pressure coefficient along the chord in different positions of the cycle and compares them with results presented by Batina (1989). The solutions in terms of aerodynamic coefficient hysteresis curves are given in Fig. 4, together with experimental data from AGARD (1982). The value of Cm is referred to the quarter-chord point. The present results are very close to those obtained by Batina (1989). Some small differences appear near shocks and the authors believe that they are due to the use of a more refined mesh in the present work. Moreover, the deviations between numerical and experimental results seem to be systematic and caused by experimental data reduction errors.

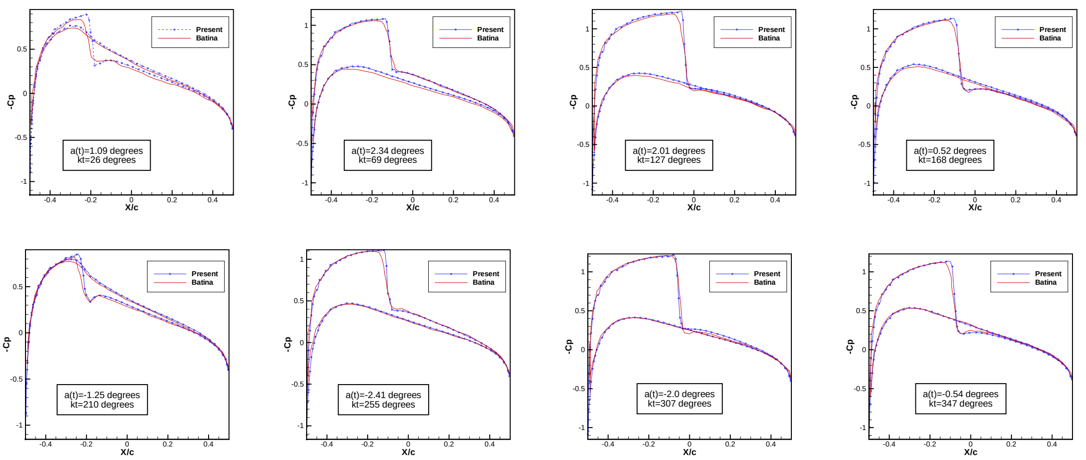


Figure 3: Comparison of instantaneous pressure distributions for the NACA 0012 airfoil in pitching motion.

Once the CFD tool was tested and proved to be a reliable one, the next step was to proceed in obtaining the unsteady responses of interest. The approach selected was to reproduce the results presented by Rausch, Batina and Young (1990) for a flat plate impulse response at $M_\infty = 0.5$. This response was obtained for pitching about the quarter-chord point, Fig. 5, and for plunging, Fig. 6. The correspondent Fourier transform, together with the results given by Rausch, Batina and Young (1990), are shown in Figs. 7 and 8.

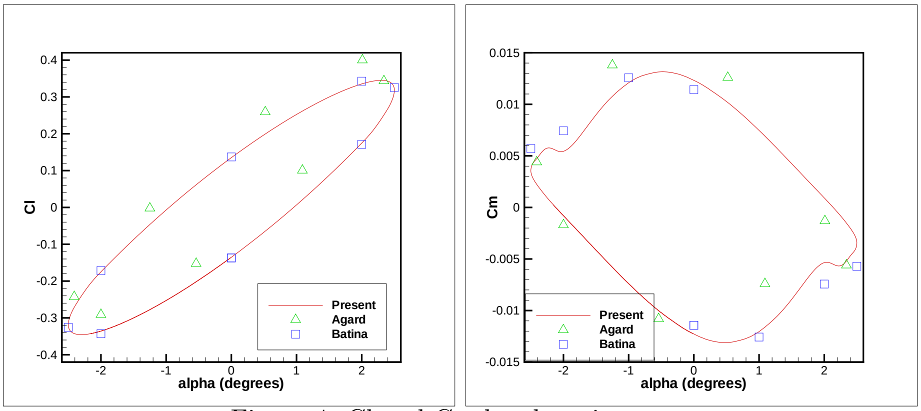


Figure 4: Cl and Cm hystheresis curves.

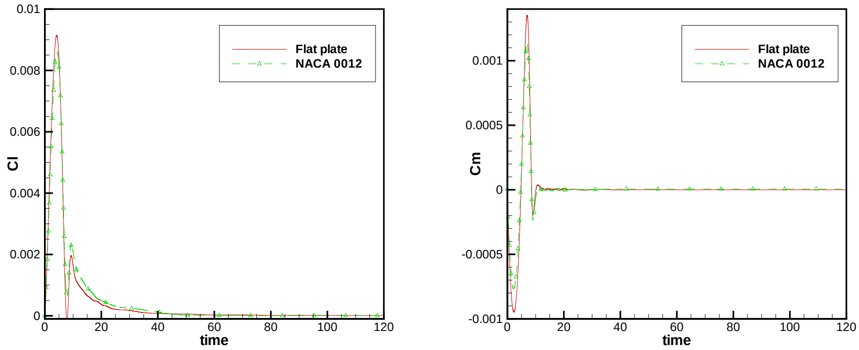


Figure 5: Cl and Cm response along time to pitching impulse excitation.

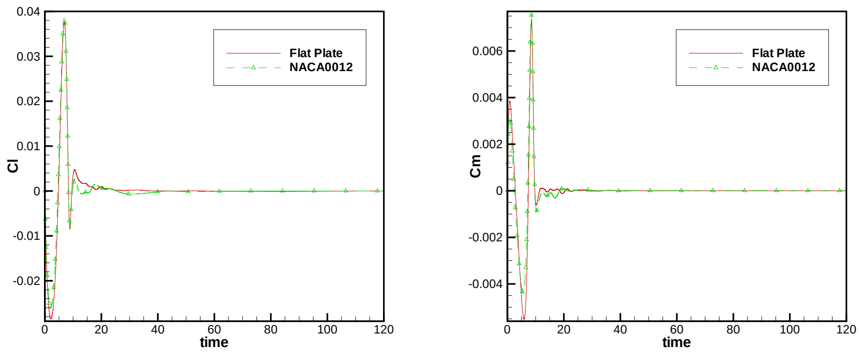


Figure 6: Cl and Cm response along time to plunging impulse excitation.

Thse results show a very good agreement between calculations performed by the present authors and the literature data. This means that the first three items which compose this methodology are completely done. Unfortunetly, to the date this work was written, the methodology has not been entierly applied and tested. However, the authors will continue this effort even more encouraged by the success obtained so far.

7 Concluding Remarks

One can see in the present results, this is an on going work since no aeroelastic problem has been really studied. Nevertheless, the preliminary results obtained so far are excellent and encourage the authors to move forward to successfully achieve the complete verification of the methodology proposed. The CFD tool developed by the CTA/IAE group has been widely tested and has proved to be a reliable source of the aerodynamic data for aeroelastic use in the subsonic regime. However, the proposed methodology aims the transonic regime, and further experiments will determine if it provides coherent results. The initial unsteady results presented here indicate that the CFD tool behaves very well in the transonic regime too.

Once validated, this methodology will provide the required capabilities to study aeroelastic problems using reduced order models (ROM) in a near future. Therefore, this work developement is a fundamental research to the evolution of aeroelastic numerical studies at CTA/IAE.

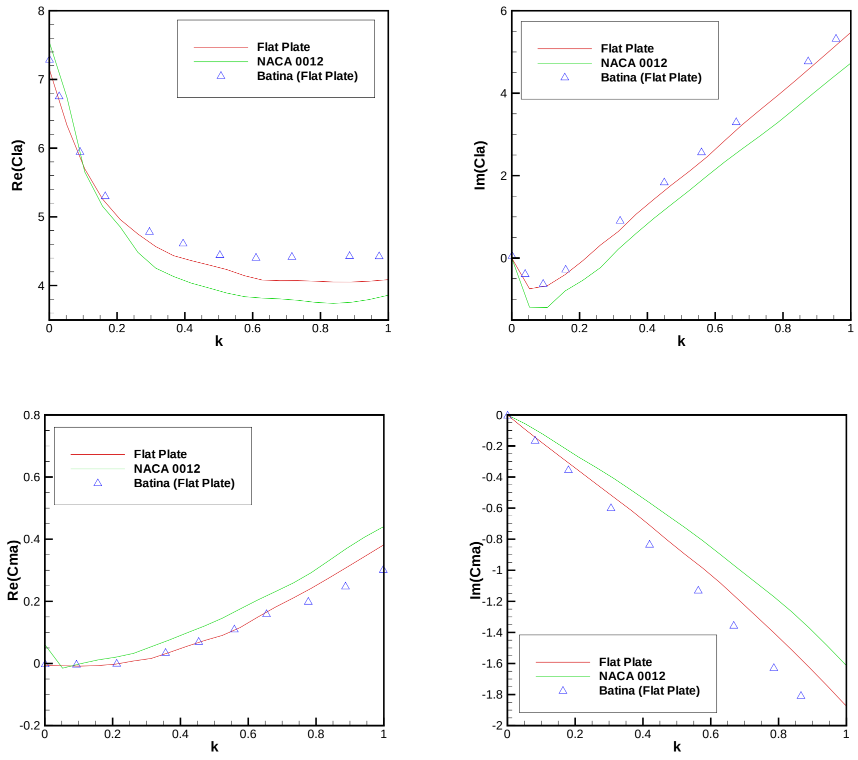


Figure 7: Cl and Cm response to pitching impluse excitation in frequency domain.

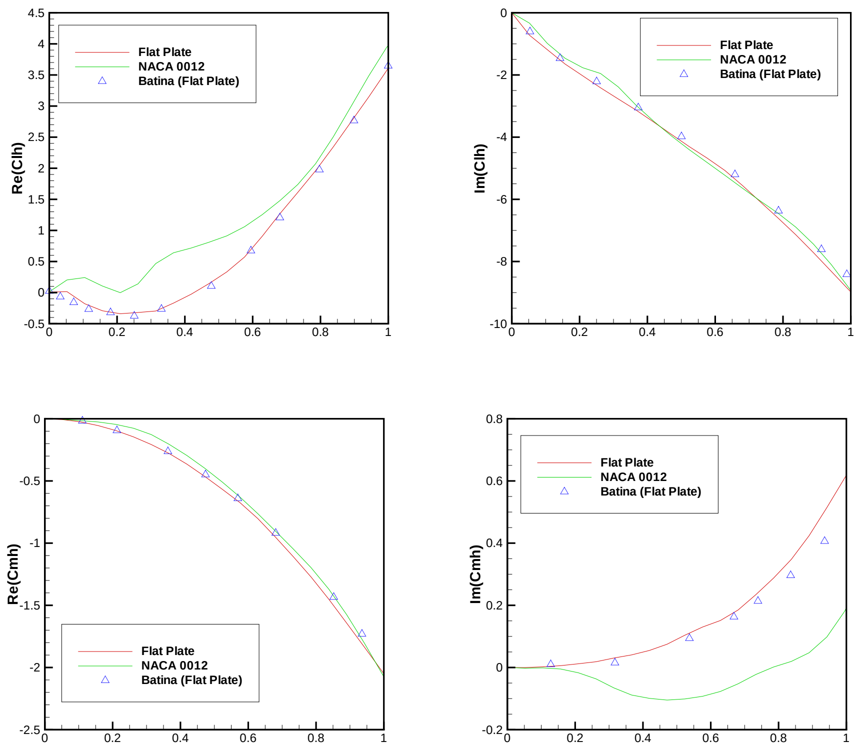


Figure 8: Cl and Cm response to plunging impluse excitation in frequency domain.

8 Acknowledgments

The authors gratefully acknowledge the partial support for this research provided by Conselho Nacional de Desenvolvimento Científico e Tecnológico, CNPq, under the Integrated Project Research Grant No.

501200/2003-7. The first author was also supported by a PIBIC scolarship under Grant No. 109339/2001-0.

9 References

AGARD Report No. 702, "Compendium of Unsteady Aerodynamic Measurments," 1982.

Azevedo, J.L.F., "Euler Solutions of Transonic Nozzle Flows," III Encontro Nacional de Ciências Térmicas - III ENCIT, Itapema, SC, Dec. 1990, pp. 243-248.

Azevedo, J.L.F., Fico, N.G.C.R., Jr. and Ortega, M.A., "Two-Dimensional and Axisymmetric Nozzle Flow Computation Using the Euler Equations," Revista Brasileira de Ciências Mecânicas, Vol. 17, No. 2, 1995, pp. 147-170.

Azevedo, J.L.F., Strauss, D. and Ferrari, M.A.S., "Viscous Multiblock Simulations of Axisymmetric Launch Vehicle Flows," AIAA Paper No. 97-2300, Proc. of 15th AIAA Applied Aerodynamics Conference, Atlanta, GA, June 1997.

Azevedo, J.L.F., "On The Development of Unstructured Grid Finite Volume Solvers for High Speed Flows," Relatório NT-075-ASE-N/92, Instituto Tecnológico de Aeronáutica, São José dos Campos, SP, Dec. 1992 (also available as Report No. 91/9, Institut für Strömungsmechanik, T.U. Braunschweig, F.R. Germany, Dec. 1992).

Bakhle, M.A. *et al.*, "Cascade Flutter Analysis with Transient Response Aerodynamics," Computers and Structures, Vol. 41, No. 5, 1991, pp. 1073-1085.

Batina, J.T., "Unsteady Euler Airfoil Solutions Using Unstructured Dynamic Meshes," AIAA Paper 89-0115, 27th Aerospace Sciences Meeting, Reno, NV, Jan. 1989.

Bigarelli, E.D.V., Mello, O.A.F. and Azevedo, J.L.F., "Three Dimensional Flow Simulations for Typical Launch Vehicles at Angle of Attack," XV Congresso Brasileiro de Engenharia Mecânica, Águas de Lindóia, SP, Nov. 1999.

Bigarelli, E.D.V. and Azevedo, J.L.F., "On Turbulence Models for 3-D Aerospace Applications," Proceedings of the 9th Brazilian Congress of Thermal Engineering and Sciences, Paper CIT02-0341, Caxambu, MG, Oct. 2002.

Bisplinghoff, H.L., Ashley, H. and Halfman, R.L., "Aeroelaticity," Adison-Wesley, Reading, Mass., 1955.

Brigham, E.O., "The Fast Fourier Transform and its Applications," Prentice Hall, Englewood Cliffs, New Jersey, 1988.

Fico, N.G.C.R., Jr. and Azevedo, J.L.F., "Numerical Investigation of Supersonic Flow in a Slotted Wind Tunnel," XV Congresso Ibero Latino-Americano sobre Métodos Computacionais para Engenharia - XV CILAME, Belo Horizonte, MG, Dec. 1994, pp. 314-322.

Fletcher, C.A.J., "Computational Techniques for Fluid Dinamics - Fundamental and General Techinques," Vol. I, Springer Verlag, Berlin, 1988a.

Fletcher, C.A.J., "Computational Techniques for Fluid Dinamics - Fundamental and General Techinques," Vol. II, Springer Verlag, Berlin, 1988b.

Hirsch, C., "Numerical Computation of Internal and External Flows - Fundamental of Numerical Discretization," Vol. I, John Wiley & Sons, 1994.

Jameson, A. and Mavriplis, D., "Finite Volume Solution of the Two-Dimensional Euler Equations on a Regular Triangular Mesh," AIAA Journal, Vol. 24, No. 4, April 1986, pp. 611-618.

Jameson, A. and Baker, T.J. "Solution of the Euler Equation for Complex Configurations," AIAA Paper 83-1929, July 1983.

Jameson, A. and Baker, T.J. "Improvements to the Aircraft Euler Method," AIAA Paper 87-0452, AIAA 25th Aerospace Sciences Meeting, Reno, NV, Jan. 1987.

Jameson, A., Schmidt, W. and Turkel, E., "Numerical Solution of the Euler equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes," AIAA Paper 81-1259, AIAA 14th Fluid and Plasma Dynamics Conference, Palo Alto, CA, June 1981.

Marques, A.N., "Simulação de Escoamentos Aeroespaciais Não Estacionários Utilizando Malhas Não Estruturadas," PIBIC-ITA Partial Activities Report, Instituto Tecnológico de Aeronáutica, São José dos Campos, SP, Brazil, Mar. 2004.

Mavriplis, D.J., "Accurate Multigrid Solution of the Euler Equations on Unstructured and Adaptative Meshes," AIAA Journal, Vol. 28, No. 2, Feb. 1990, pp.213-221.

Mohr, R.W., Batina, J.T. and Yang, H.T.Y., "Mach Number Effects on Transonic Aeroelastic Forces and Flutter Characteristics," Journal of Aircraft, Vol. 26, No. 11, 1989, pp. 1038-1046.

Oliveira, L.C., "A State-Space Aeroelastic Analysis Methodology Using Computational Aerodynamics Techniques," Master Thesis, instituto tecnológico de Aeronáutica, São José dos Campos, SP, Brazil, Sept. 1993 (in Portuguese, original title is "Uma Metodologia de Análise Aeroelástica com Variáveis de Estado Utilizando Técnicas de Aerodinâmica Computacional").

Pulliam, T.H., "Artificial Dissipation Models for the Euler Equations," AIAA Journal, Vol. 24, No. 12, Dec. 1986, pp. 1931-1940.

Rausch, R.D., Batina, J.T. and Yang, H.T., "Euler Flutter Analysis of Airfoil Using Unstructured Dynamic Meshes," Journal of Aircraft, Vol. 27, No. 5, May 1990, pp. 436-443.

Simões, C.F.C. and Azevedo, J.L.F., "Unsteady Airfoil Inviscid Flow Simulations Using Unstructured Dynamic Meshes," 15 Congresso Brasileiro de Engenharia Mecânica, Bauru, SP, Brazil, 1999.

Tijdeman, H., "Investigation of tbe Transonic Flow Around Oscillating Airfoils," NLR-TR-77090U, National Aerospace Laboratory, The Netherlands, Dec. 1977.

10 Responsibility Notice

The authors are the only responsible for the printed material included in this paper.