NUMERICAL SOLUTION OF THE EULER EQUATIONS FOR THE REPRESENTATION OF TRANSONIC FLOWS ON AIRFOILS

Elizangela Camilo - elcamilo@sc.usp.br
Flávio Donizeti Marques - fmarques@sc.usp.br
Universidade de São Paulo, USP
Escola de Engenharia de São Carlos, EESC
Av. Trabalhador Sancarlense, 400
13566-590 - São Carlos - SP - Brasil

Abstract. *The study of aerodynamic modeling methods in transonic flow regimes has a great importance to the aeronautical engineering. Nonlinear features due to compressibility effects and shock waves appearance represent a major challenge on the treatment of transonic aerodynamics. Such nonlinear effects present a great influence on aerodynamic performance, as well as they are responsable for harmful aeroelastic response phenomena in aircraft. Equations for transonic flows can be obtained from the basic fluid mechanic theory. However, only numerical methods are able to attain practical solutions for those set of partial differential equations in the present moment. For the specific case of treating transonic flow problems the nonlinear Euler equations provide an appropriate set of partial differential equations to capture nonlinear effects of typical compressible flows, despite of not accounting for viscous flows effects. The aim of this work is to develop a computational routine to the numerical solution of transonic flows around airfoils. A finite difference C-type structured mesh has been used to discretize the flow around a NACA0012 airfoil. The methodology for numerical solution has been based on the explicit MacCormack method of second order in time and space. Artificial dissipation with nonlinear coefficients has incorporated to the method. The steady transonic flow around the NACA0012 airfoil numerical solution is assessed and the main flow properties are presented. Shock waves structure can also be observed by means of the Mach number contours around the airfoil. Pressure distributions on upper and lower surfaces for different flow conditions are also shown, thereby allowing the observation of the abrupt pressure change effects due to shock waves. The present work results match well with the solutions obtained in other computational codes used for the same problem that are presented in the literature.*

keywords: Euler equations, CFD methods, transonic flows.

1. INTRODUCTION

The problems associated with compressibility effects on high performance aircraft have contributed significantly to increase the difficulties in aeronautical design. A flow regime in which compressibility effects introduce a great deal of difficulties is the transonic one. Major challenge on the treatment of transonic flows is due to nonlinear behavior. The dominant source of nonlinearity corresponds to the formation of shock waves and, for many cases, viscous effects can be neglected.

The Euler equations have been used for the representation of transonic flows to capture mixed flow effect (subsonic and supersonic) and the consequent shock waves formation. The mathematical theory for equations of this kind is still not developed to allow the attainment of analytical solutions in arbitrary regions and general boundary conditions. The most appropriate alternative is to apply a numerical method. The major current effort in computational methods for aerodynamics is the development of tools using methods of Computational Fluids Dynamics (CFD) (Anderson, 1995, Hirsch, 1988a, Hirsch, 1988b, Fletcher, 1992a, Fletcher, 1992b).

The interest in CFD for aeronautical applications is to acquire trustworthy and practical solutions of aerodynamic models. Great advances are already being reached and applications of CFD in the aeronautical industry start to be more usual.

The solution of the Euler equations for CFD methods has been proposed for a significant number of researchers (Nixon, 1989, Anderson, 1995). Shock waves formation and consequent associated nonlinear behavior can be successfully treated by CFD methods. Another field in which CFD methods present increasing interest is in the treatment of aeroelastic problems (Bisplinghoff, 1996). The so-called *computational aeroelasticity* (Bennett and Edwards, 1998) is a term that defines a field of aeroelasticity theory that relates the integration of computational tools for solving aeroelastic problems in general. The current trend in the use of CFD methods enables unsteady aerodynamic modeling, while the finite element methods have been the most appropriate to represent structural-dynamics problems. Therefore, a typical computational aeroelasticity model comprises the coupling of structural-dynamics and unsteady aerodynamic models.

In this work the first step towards the development of aerodynamic models to apply in aeroelasticity (Camilo, 2003) is presented. In the context of steady aerodynamic modeling methods for the transonic flow regime, the aim of this work is to implement a CFD method for the numerical solution of the nonlinear Euler equations. The model will serve to ensure that the main steady transonic flow phenomena will be captured. Then, futher developments in unsteady transonic aerodynamic modeling can be attained for aeroelastic analysis. The CFD method has been based on the explicit MacCormack method of second order in time and space with artificial dissipation added to the method (Hirsch, 1988b). A finite difference C-type mesh is used to discretize the flow field around a NACA 0012 (NACA - National Adivisory Comittee of Aeronautics) airfoil. A complete analysis of the nonlinear flow phenomena is presented.

2. EQUATIONS FOR TWO-DIMENSIONAL COMPRESSIBLE INVISCID FLOWS IN TRANSFORMED COORDINATES

For non-uniform or curvilinear meshes the discretization of the equation can be performed after a transformation from the physical space (x, y, t) to a computational space (ξ, η, τ). The relations between the two spaces are defined through the coordinate transformation formulas. These transformation represent a mapping from the physical space to the computational space. Therefore, the Euler equations can be written in curvilinear coordinates (Nixon, 1989).

The conservative formulation of the nonlinear Euler equations in transformed coordinate system can be written as:

$$\frac{\partial \bar{\mathbf{Q}}}{\partial \tau} + \frac{\partial \bar{\mathbf{F}}}{\partial \xi} + \frac{\partial \bar{\mathbf{G}}}{\partial \eta} = \mathbf{0}, \tag{1}$$

where $\bar{\mathbf{Q}}$ denotes the vector of conservative variables, $\bar{\mathbf{F}}$ and $\bar{\mathbf{G}}$ denote the convective fluxes, that is:

$$\bar{\mathbf{Q}} = J^{-1} \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho e_t \end{bmatrix}, \; \bar{\mathbf{F}} = J^{-1} \begin{bmatrix} \rho u \\ \rho u U + \xi_x p \\ \rho v U + \xi_y p \\ U(\rho e_t + p) - \xi_t p \end{bmatrix}, \; \bar{\mathbf{G}} = J^{-1} \begin{bmatrix} \rho V \\ \rho u V + \eta_x p \\ \rho v V + \eta_y p \\ V(\rho e_t + p) - \eta_t p \end{bmatrix}, \tag{2}$$

where u and v are the two components of the cartesian velocity, e_t the specific total energy, p the pressure, J is the jacobian defined by:

$$J = (\xi_x \eta_y - \eta_x \xi_y) = (x_\xi y_\eta - y_\xi x_\eta)^{-1}, \tag{3}$$

and U and V are the contravariant velocities given by:

$$U = \xi_t + \xi_x u + \xi_y v, \qquad V = \eta_t + \eta_x u + \eta_y v. \tag{4}$$

3. NUMERICAL SOLUTION OF THE EULER EQUATIONS

3.1 General Description of the Explicit Method

The second-order explicit MacCormack method is derived from the basic explicit Lax-Wendroff scheme (Hirsch, 1988b). They can be interpreted as space-centred discretizations, first-order discretization in the time and an additional dissipative term to guarantee second-order accuracy.

3.2 Explicit MacCormack Scheme with Artificial Dissipation

MacCormack's scheme combines forward and backward differences in the space discretization with predictor and corrector steps. The artificial dissipation terms are generally added into both predictor and corrector steps (Hirsch, 1988b), that is:

Predictor:

$$\bar{\mathbf{Q}}^{m}_{j,k} = \bar{\mathbf{Q}}^{n}_{j,k} - \tau_\xi(\bar{\mathbf{F}}^{n}_{j+1,k} - \bar{\mathbf{F}}^{n}_{j,k}) - \tau_\eta(\bar{\mathbf{G}}^{n}_{j,k+1} - \bar{\mathbf{G}}^{n}_{j,k}) + \mathbf{D}^{n}_{j,k}, \tag{5}$$

Corrector:

$$\bar{\mathbf{Q}}^{m+1}_{j,k} = \bar{\mathbf{Q}}^{n}_{j,k} - \tau_\xi(\bar{\mathbf{F}}^{m}_{j,k} - \bar{\mathbf{F}}^{m}_{j-1,k}) - \tau_\eta(\bar{\mathbf{G}}^{m}_{j,k} - \bar{\mathbf{G}}^{m}_{j,k-1}) + \mathbf{D}^{m}_{j,k}. \tag{6}$$

The solution for the time $n+1$ is given by:

$$\bar{\mathbf{Q}}^{n+1}_{j,k} = \frac{1}{2}(\bar{\mathbf{Q}}^{m} + \bar{\mathbf{Q}}^{m+1}). \tag{7}$$

Jameson et al. (1981) and Pulliam (1986) have suggested the construction of a dissipative operator with the following form:

$$\mathbf{D}_{j,k} = d_{j+\frac{1}{2},k} - d_{j-\frac{1}{2},k} + d_{j,k+\frac{1}{2}} - d_{j,k-\frac{1}{2}}, \tag{8}$$

where the dissipative fluxes are defined by:

$$d_{j+\frac{1}{2},k} = \alpha_{j+\frac{1}{2},k}\big(d^{(2)}_{j+\frac{1}{2},k} - d^{(4)}_{j+\frac{1}{2},k}\big), \tag{9}$$

where

$$\alpha_{j+\frac{1}{2},k} = \frac{1}{2}\big(\sigma_{j+1,k}J^{-1}_{j+1,k} + \sigma_{j,k}J^{-1}_{j,k}\big), \tag{10}$$

with

$$\sigma_{j,k} = |U| + a\sqrt{\xi_x^2 + \xi_y^2} + |V| + a\sqrt{\eta_x^2 + \eta_y^2}. \tag{11}$$

The use of second and fourth differences operators also follows the ideas of Pulliam (1986) for the control of the nonlinear instability, and they are given by:

$$d^{(2)}_{j+\frac{1}{2},k} = \epsilon^{(2)}_{j+\frac{1}{2},k}(\mathbf{Q}_{j+1,k} - \mathbf{Q}_{j,k}), \tag{12}$$

$$d^{(4)}_{j+\frac{1}{2},k} = \epsilon^{(4)}_{j+\frac{1}{2},k}(\mathbf{Q}_{j+2,k} - 3\mathbf{Q}_{j+1,k} + 3\mathbf{Q}_{j,k} - \mathbf{Q}_{j-1,k}), \tag{13}$$

with

$$\epsilon^{(2)}_{j+\frac{1}{2},k} = K_2\Delta t_{j,k} max(\Gamma_{j+2,k}, \Gamma_{j+1,k}, \Gamma_{j,k}, \Gamma_{j-1,k}), \tag{14}$$

$$\epsilon^{(4)}_{j+\frac{1}{2},k} = max\Big[0, \Big(K_4\Delta t - \epsilon^{(2)}_{j+\frac{1}{2},k}\Big)\Big], \tag{15}$$

and

$$\Gamma_{j,k} = \frac{|p_{j+1,k} - 2p_{j,k} + p_{j-1,k}|}{|p_{j+1,k} + 2p_{j,k} + p_{j-1,k}|}, \tag{16}$$

where K_2 and K_4 are real parameters.

4. SOLUTION OF THE STEADY TRANSONIC FLOWS AROUND AN AIRFOIL

A C-type structured grid has been used around the airfoil. The Figure 1 presents an illustration of the mesh closer to the airfoil. For all calculations, the initial conditions have been especified with the freestream values.

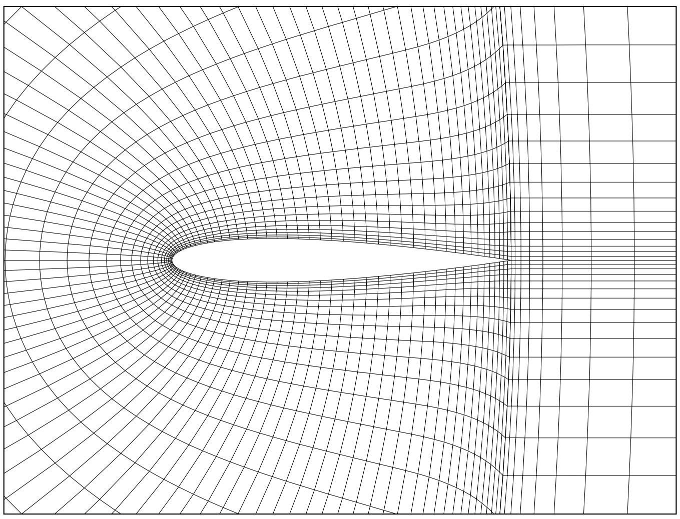

Figure 1: C-type structured mesh closer to the NACA0012 airfoil .

4.1 Boundary Conditions

External boundaries have the limits located far enough from the main flow region, so that the influence of the flow disturbances does not affect the freestream values. Therefore the inlet and outlet boundaries are specified with freestream values.

Airfoil boundaries must be specified as solid wall boundary conditions. Considering the contravariant velocities U and V given by Eqs. (4), the following expressions for the Cartesian velocity in the solid wall (Hirsch, 1988b) are attained :

$$u_b = \frac{\eta_y U_b}{\eta_x^2 + \eta_y^2}, \tag{17}$$

$$v_b = \frac{\eta_x U_b}{\eta_x^2 + \eta_y^2}, \tag{18}$$

where η_x and η_y are the normal vector components to the boundary surface and the subscript b indicates that the variable is at solid boundary.

The density and pressure are approached by zero-order extrapolation methods:

$$\rho_b = \rho_2, \tag{19}$$

$$p_b = p_2. \tag{20}$$

4.2 Steady Transonic flow around the NACA0012 Airfoil

The results obtained here are compared with the data presented by Oliveira (1993). Lift and drag coefficients, the Mach number contours and pressure distributions on upper and lower surfaces of the NACA0012 airfoil have been presented. Four cases in which different incidence angle and Mach number have been considered. Table 1 presents the values used in each case.

For the numerical solution the CFL number was equal to 0.9 and the artificial dissipation parameters were: $k_2 = \frac{1}{4}$ and $k_4 = \frac{1}{100}$.

Table 1: Parameters used in the cases for the flow solution around the NACA0012 airfoils.

	Case 1	Case 2	Case 3	Case 4
M_∞	0,63	0,8	0,8	0,85
α (degrees)	2	0	1,25	1

Table 2 presents a comparison between the lift and drag coefficients obtained with the present methodology and with the ones extracted from Oliveira (1993).

Figures 2 to 5 show the Mach number contours revealing the mixing condition of the subsonic and supersonic flow around of the airfoil. In the case 1 the airfoil is exposed to a condition of relatively low Mach number, where the shock wave structure can not be observed. In the other cases (2 to 4) the influence of the Mach number is clear in the formation of the shock wave.

Figures 6 to 9 present the pressure distributions around of the NACA0012 airfoil for the cases presented in Tab.1. The results are also compared with the ones obtained by Kroll and Jain (1987). Kroll and Jain (1987) have used finite volume methods to the numerical solution of the Euler equations.

Table 2: Aerodynamic coefficients for the NACA0012 airfoil. (Pres.- Present work; Ref.- Oliveira (1993))

	Case 1		Case 2		Case 3		Case 4	
	Pres.	Ref.	Pres.	Ref.	Pres.	Ref.	Pres.	Ref.
C_L	$0,3262$	$0,3275$	$0,0000$	$0,0020$	$0,3348$	$0,3509$	$0,3845$	$0,3567$
C_D	$0,0065$	$0,0033$	$0,0104$	$0,0103$	$0,0263$	$0,0243$	$0,0628$	$0,0585$

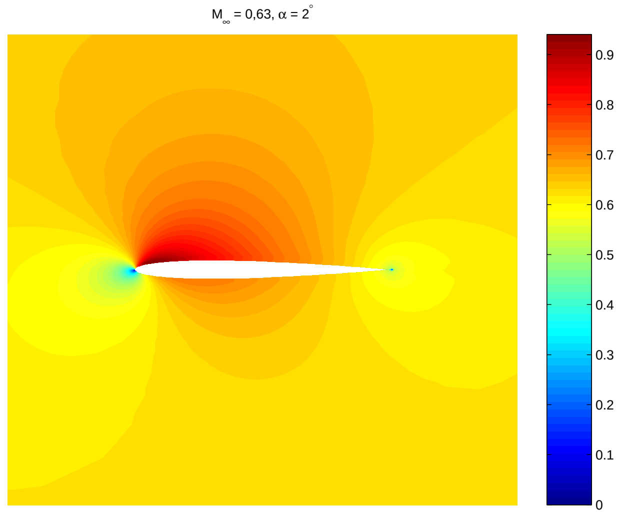

Figure 2: Mach number contours around of the NACA0012 airfoil for the **case 1** conditions.

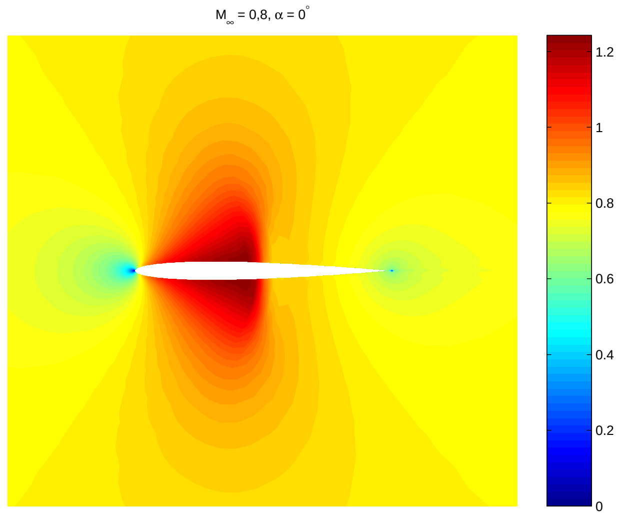

Figure 3: Mach number contours around of the NACA0012 airfoil for the **case 2** conditions.

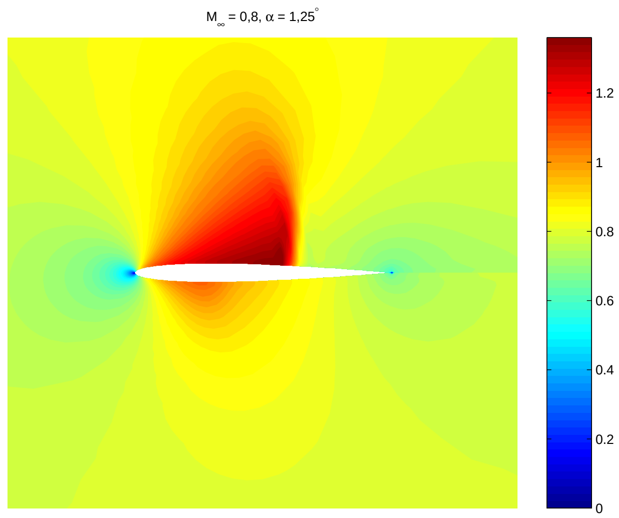

Figure 4: Mach number contours around of the NACA0012 airfoil for the **case 3** conditions.

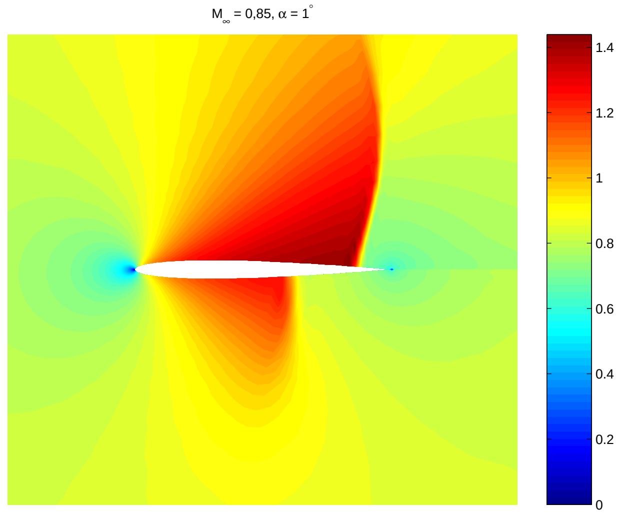

Figure 5: Mach number contours around of the NACA0012 airfoil for the **case 4** conditions.

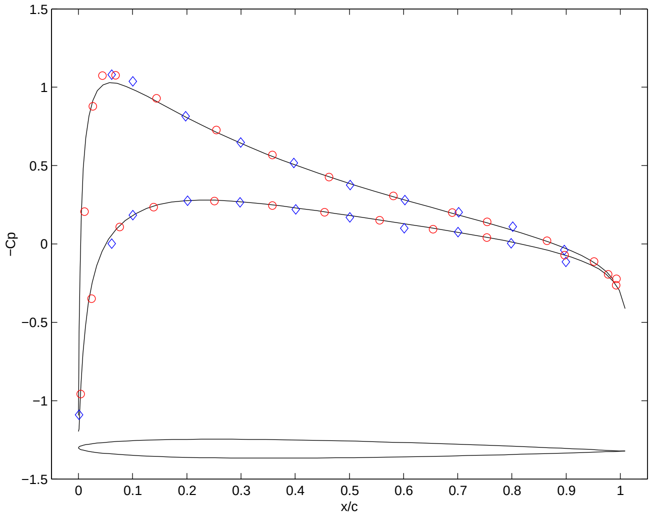

Figure 6: Pressure distributions for the NACA0012 airfoil, **case 1**. Solid lines - results of the present work; ○ - extracted from the Oliveira (1993); ◇ - extracted from the Kroll and Jain (1987).

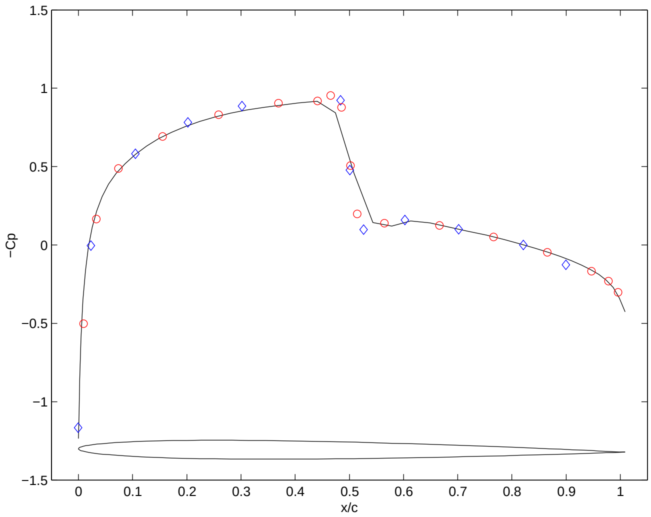

Figure 7: Pressure distributions for the NACA0012 airfoil, **case 2**. Solid lines - results of the present work; ○ - extracted from the Oliveira (1993); ◇ - extracted from the Kroll and Jain (1987).

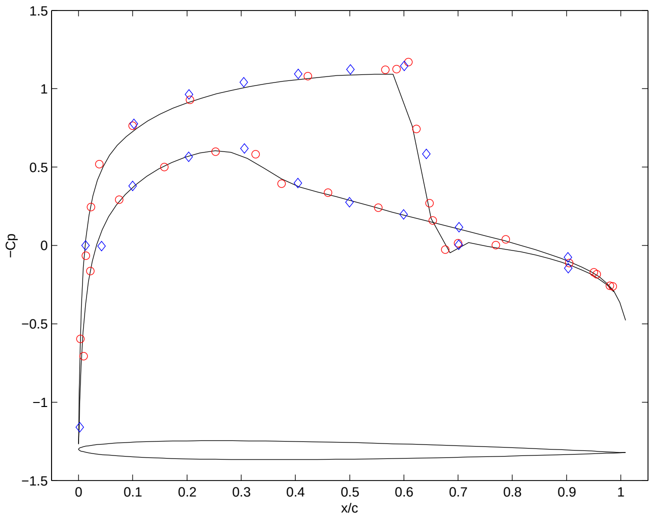

Figure 8: Pressure distributions for the NACA0012 airfoil, **case 3**. Solid lines - results of the present work; ○ - extracted from the Oliveira (1993); ⋄ - extracted from the Kroll and Jain (1987).

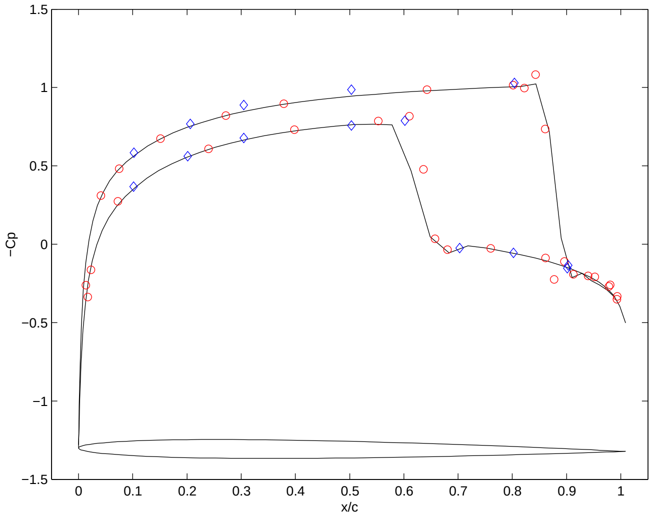

Figure 9: Pressure distributions for the NACA0012 airfoil, **case 4**. Solid lines - results of the present work; ○ - extracted from the Oliveira (1993); ⋄ - extracted from the Kroll and Jain (1987).

5. CONCLUSIONS

This work has presented the implementation of the explicit MacCormack method, with addition of artificial dissipation with nonlinear coefficients, for the solution of the two dimensional nonlinear steady Euler equations. The procedure aims to be the first steps towards the development of models for computational aeroelasticity. Four cases have been considered, where the Mach number and incidence angle varied inside of a range that guarantees no flow separation. The analysis of the results have been based on the Mach number contours for the discretized field, the respective steady pressure distributions and aerodynamic coefficients calculation. The formation of shock waves in agreement with the expected physical effect in compressible flows have been observed. All the cases agree well with the solution obtained in other computational codes presented in the literature that have been used for the same problem. For continuity of the work the solution of the unsteady Euler equations in transonic flow can be based on the prescribed oscillatory movements for a airfoil one. The development of this method can lead to future implementation of algorithms for simultaneous solution of the aerodynamic and structural dynamics equations in the context of the computational aeroelasticity.

Acknowledgements

The authors would like to acknowledge the financial support from the Conselho Nacional de Desenvolvimento Científico e Tecnológico - CNPq and from the Fundação de Amparo à Pesquisa de Estado de São Paulo - FAPESP.

References

Anderson, J. D. Jr., ”Computational fluid dynamics - the basics with applications”, McGraw-Hill International Editions, 1995.

Bennett, R. M. and Edwards, J. W., ”An overview of recent developments in computational aeroelasticity”, AIAA Journal, pp. 1-14, Jun 1998.

Bisplinghoff, R. L. & Ashley,H. and Halfman, R. L., ”Aeroelasticity”, Dover - New York, 1996.

Camilo, E., ”Solução numérica das equações de Euler para representação do escoamento transônico em aerofólios”, São Carlos. 72p. Dissertação (Mestrado) - Escola de Engenharia de São Carlos, Universidade de São Paulo, 2003.

Fletcher, C. A. J., ”Computational techniques for fluid dynamics: fundamental and general techniques”, Springer-Verlag, vol. 1, 2 ed., 1992a.

Fletcher, C. A. J., ”Computational techniques for fluid dynamics: specific techniques for different flow categories”, Springer-Verlag, vol. 2, 2 ed., 1992b.

Hirsch, C., ”Numerical computation of internal and external flows: fundamentals of numerical discretization”, John Wiley & Sons Ltd., vol. 1, 1988a.

Hirsch, C., ”Numerical computation of internal and external flows: computational methods for inviscid and viscous flows”, John Wiley & Sons Ltd., vol. 2, 1988b.

Jameson, A. and Schmidt, W. & Turkel, E., ”Numerical simulation of the Euler equation by finite volume using Runge-Kutta time stepping schemes”, AIAA Paper, pp. 81-1259, 1981.

Kroll, N. e Jain, R.K., “Solutions of the two-dimensional Euler equations - experience with finite volume code”, DFVLR-FB 87-41, 1987.

Nixon, A., “Unsteady transonic aerodynamics”, Progress in Astronautics and Aeronautics, 1989. (American Institute of Aeronautics and Astronautics, vol. 120).

Oliveira, L. C., ”Uma metodologia de análise aeroelástica com variáveis de estado utilizando técnicas de aerodinâmica computacional”, São José dos Campos. 100p. Dissertação (Mestrado) - Instituto Tecnológico de Aeronáutica, 1993.

Pulliam, T. H., ”Artificial dissipation models for the Euler equations”, AIAA Journal, vol.24, n. 12, pp. 1931-1940, Dec 1986.