International Journal of Mechanics and Energy (IJME)
Vol. 6, Issue 1, 2018, ISSN: 2286-5845

# Simulation of transonic flow around the aerodynamic airfoils and wings (Application to UAV)

Rachid Renane*, Rachid.ALLOUCHE*, Nadir Bekka*, Abdelkader Nour**

*(*) Laboratory of aeronautical sciences IAES University of Blida1, Algeria*

*(**) Laboratory of engine dynamiqcs & vibroacoustics UMBB, Algéria*

**Abstract:**
The flight regime of modern transport aircraft, is at transonic speeds, in terms of Mach number $M_\infty$ = 0.75-0.85 and at very high Reynolds numbers. The transonic flow occurs when the two regimes of subsonic and supersonic flows occur in the same local flow field. It is characterized by the development of a supersonic pocket, delimited by the wall on the one hand and by the sonic line on the upper surface of the wing on the other hand. The appearance of a supersonic pocket on the under-surface of the wing is also possible; all depends on the incidence angle and the geometrical form of the wing. This work is devoted to the evaluation and the validation of the numerical results resulting from transonic calculation of flow around a NACA0012 aerofoil, then around a wing 3d having the same basic airfoil. All calculations are executed by using the turbulence models of two equations k-ε, k-ω and SST k-w.

**Key words:** Transonic flow, Numerical simulation, Aerofoil NACA0012, Mach wave, unmanned aircraft vehicle (UAV)

## 1. Introduction

The development of drones is deviated more to the high speeds field and transonic Mach numbers. That is particularly obvious for the interest of development of "HALE" aircraft type, the evaluation of transonic flying wings, design of drones at high speed and improvement of the capacities of UAV (unmanned aircraft vehicle) operations. Unfortunately, even the preliminary design stapes of the transonic configurations are complicated because the absence of appropriate airfoils and their data performances, moreover the majority of the existing high speed airfoils were conceived for civil aircrafts applications and their airfoil of specific mission. For that, the design of new transonic wings is a practically inevitable choice for the significant improvement of the aerodynamic performances and to fill between the design requirements. in this work, we simulated a transonic flow of air around a wing with symmetrical airfoil of NACA0012 type on account of validation.

For 2D case; a house code was used for the generation of grid around airfoil by the "algebraic" method, for the wing "3D case", we used the Ansys software for grid generation and simulation of aerodynamic performances with kw, ke and SST turbulence models.

## 2. Transsonic mode

The transonic mode met in the cruise flight of transport aircraft, and in the flight operation of the military aircrafts and UAV is characterized by a upstream Mach number slightly inferior of the unit, what will shout a supersonic pocket of flow limited by a sonic line around the geometry in question. The latter generally ends by a shock wave which decelerate the flow towards the subsonic conditions. See figures1 and 2.

* **Corresponding author: Rachid Renane**
E-mail: r.renane@yahoo.fr

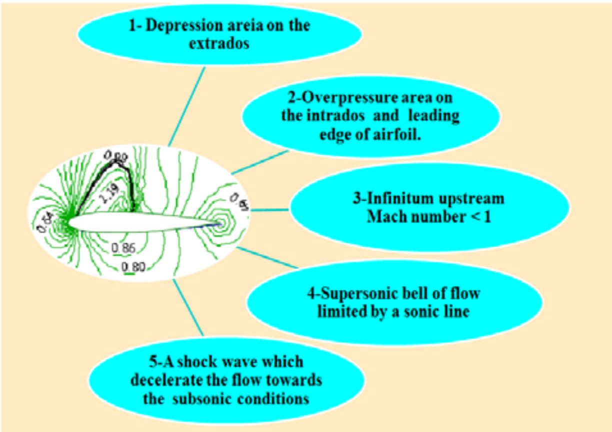



**Fig.1 Characteristics of nonviscous transonic flow around wing airfoil.**

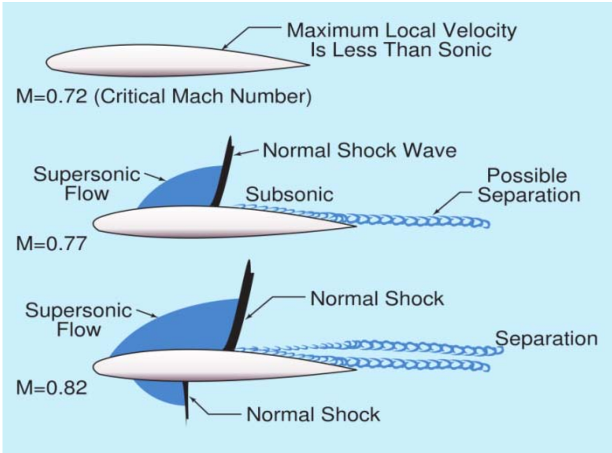



**Fig. 2 Shock waves progression on the airfoil surface of wing according to Mach number [8, W.H. Mason]**

## 3. Grid generation

The procedure of the numerical solution of Euler equations is uncoupled by grid generation. The automatic grid generation consists in generating the automatic grid generation consists in generating a system of curvilinear coordinated which determines the field of calculation in which all the equations of physical problem will be expressed. The algebraic technique used in our study suggested by Rebert (1971) and modifies by Eissemann (1979), produces a direct functional description of the transformation between the computational and physical fields. A program of grid generation around the profile of wing was establishes. Several cases can be represented by using this last, see figures 3 and 4.

## 4. Mathematical models

For this study, calculations are executed by using the Ansys code, where several models of turbulence are available in this code, the models with one and two transport equations use partial derivative equations to connect the fluctuations of flow to the average sizes of variables.

We limit as an example to present thereafter the k-w model. The K-ε model and SST- Model are respectively detailed in reference [5] and reference [7].

*4.1 K-w model.*

The K-ω model is a model with two transport equations. The equations to be solved are the equation of turbulent kinetic energy K and the specific dissipation rate ω. The turbulent viscosity is expressed by $\upsilon_t = C_\mu \dfrac{k}{\omega}$ and the transport equations are illustrated by the following equations:

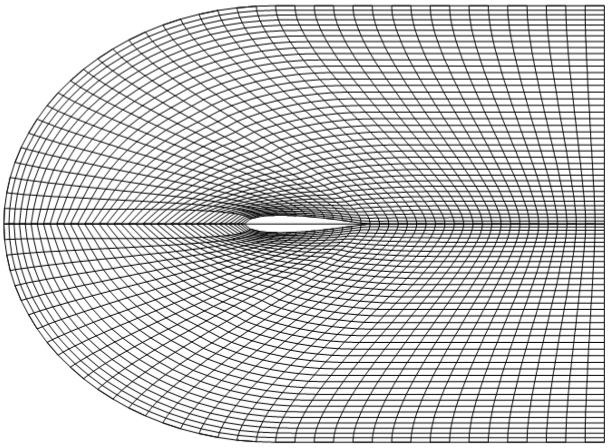


**Fig.3 Grid in C around the NACA 0012 airfoil.**

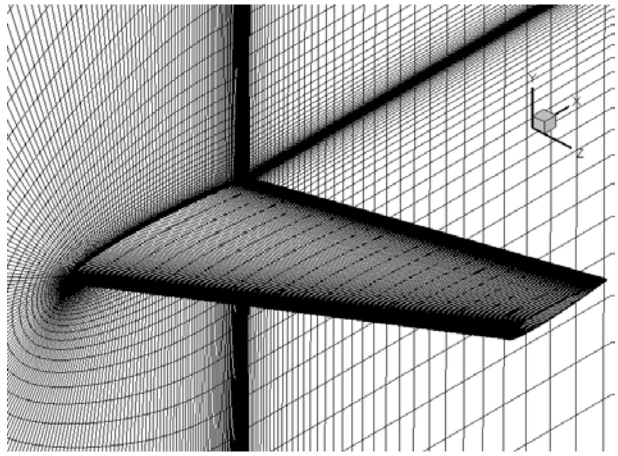


**Fig.4 Grid (3D) around the M6 wing.**

$$\frac{\partial}{\partial t}\left(\bar{\rho}k\right)+\frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{u}_j k\right)= \bar{\rho}P-\bar{\rho}\omega k+\frac{\partial}{\partial x_j}\left[\left(\bar{\mu}+\frac{\bar{\mu}_t}{\sigma}\right)\frac{\partial k}{\partial x_j}\right] \quad (1)$$

$$\frac{\partial}{\partial t}\left(\bar{\rho}\omega\right)+\frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{u}_j\omega\right)= C_{\omega1}\frac{\bar{\rho}P\omega}{k}-C_{\omega2}\bar{\rho}\omega^2+\frac{\partial}{\partial x_j}\left[\left(\bar{\mu}+\frac{\bar{\mu}_t}{\sigma_\varepsilon}\right)\frac{\partial \omega}{\partial x_j}\right] \quad (2)$$

With the constants [6]:

$$C_\mu = 0.09, C_{\omega1} = 0.555$$

$$C_{\omega1} = 0.833, P_{rk} = 2.0 \; et \; P_{r\omega} = 2.0$$

The conditions of K and ω on the wall are:

$$k = 0 \quad for \quad y = 0$$

$$\omega = 7.2\frac{\upsilon}{y^2} \quad for \quad y = y_1$$

Where y1 is the normal distance from the wall to the center of the first mesh. To have accurate solutions, the center of the first mesh must be positioned more close to the wall. This model requires a very fine grid in the vicinity of solids surface. A non-dimensioned distance near to the unit is generally recommended y+=1.

*4.2 Boundary conditions*

We consider that the initial field is uniform obtained from the upstream flow conditions. The boundary conditions are treated as follows:

• Adhesion conditions and adiabatic wall condition are imposed at the wall.

• Input / output boundary conditions type are imposed on the rest of field.

## 5. Results and comments

In order to show the influence of the upstream Mach number and the incidence angle on the establishment of the shock wave and supersonic pocket and the lift coefficient, we present the following results:

Figures 5 and 6 show the influence of upstream Mach number with the incidence angle on the pressure distribution for Mach = 0.75 and alpha = 2 deg, we note that the Cp is positive to the leading edge, so it was a overpressure, it becomes negative on the extrados, so there is a depression followed by an acceleration of flow, a sudden increase in Cp caused by the creation of the shock wave that goes back the flow to the upstream conditions. A good agreement is observed with the references [ 1,2,3,4 ],

figures 7 and 8 show the influence of upstream Mach number with the incidence angle on the

local Mach number distribution, the breakpoints are moving according to the different incidences, and the supersonic pocket widens, the sonic line is close to the edge, and the number of supersonic points on the extrados inside of the supersonic cloche increases and the shock wave is more intense and closer to the trailing edge.

Figures 9 and 10 show respectively the iso-Mach map around ONERA M6 wing for a Mach $M_\infty$=0.799 and an incidence angle $\alpha$=2.257°; and iso-Mach lines in the various stations along the wing span z/b = 0.5 and z/b = 0.75, we see clearly a Mach wave and supersonic cloche around the wing on the different stations.

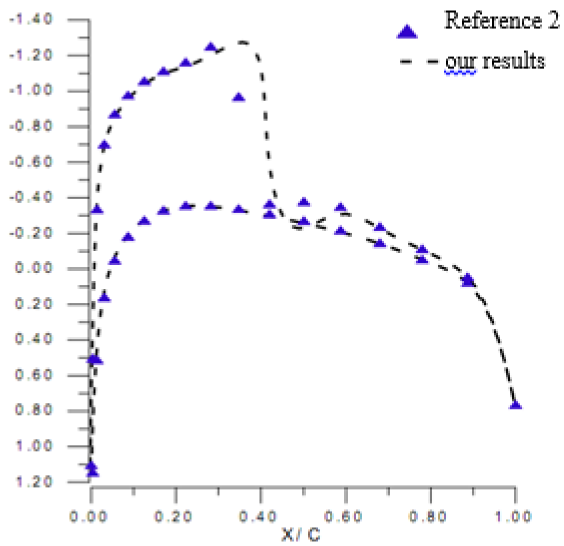
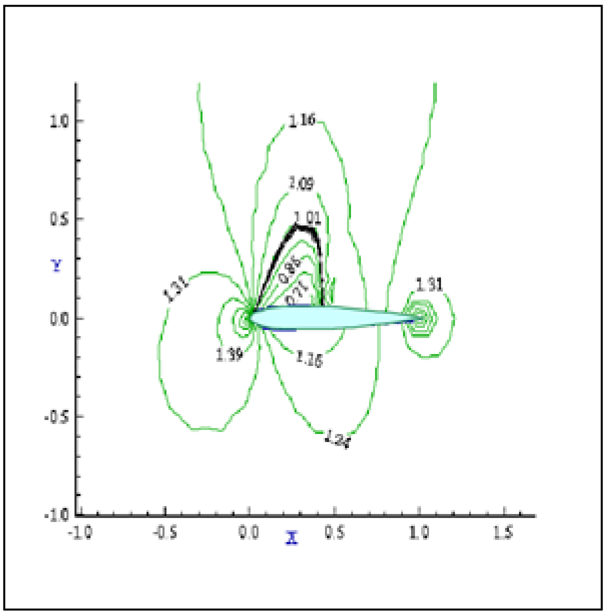



**Fig.6 Distribution of pressure coefficient on the NACA0012 airfoil surface.**




**ig.5 Iso-pressure lignes for Mach=0.75 & Alpha =2°.**

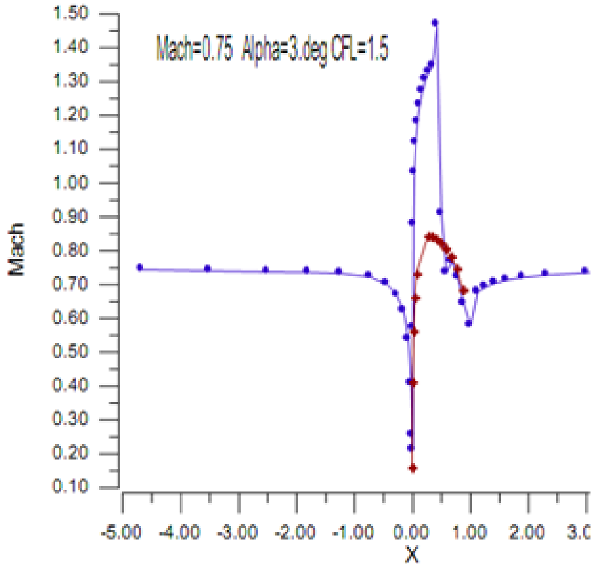



**Fig.7 Influence of upstream Mach number with the incidence angle on the local Mach number distribution.**

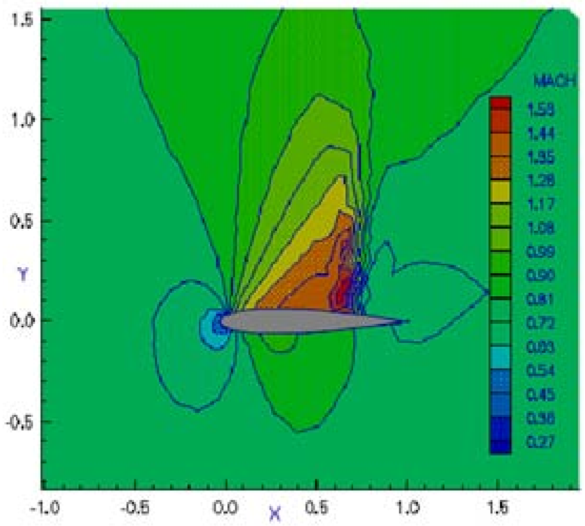



**Fig.8 Distribution of iso-mach lignes around the NACA0012 airfoil for MACH=0.8 & ALPHA=3.Deg**

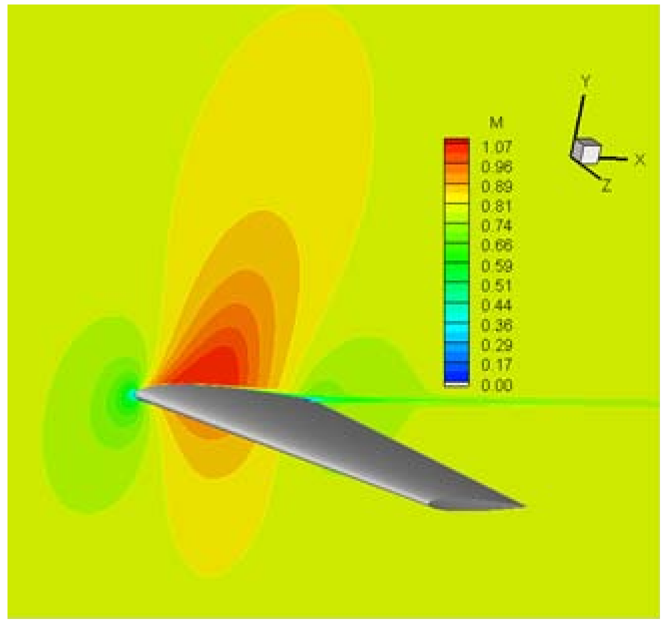



**Fig.9 Iso-Mach map for $M_\infty$ =0.799; $\alpha_c$ =2.257°; $Re_\infty$ =9.10$^6$ (SST k-w)**

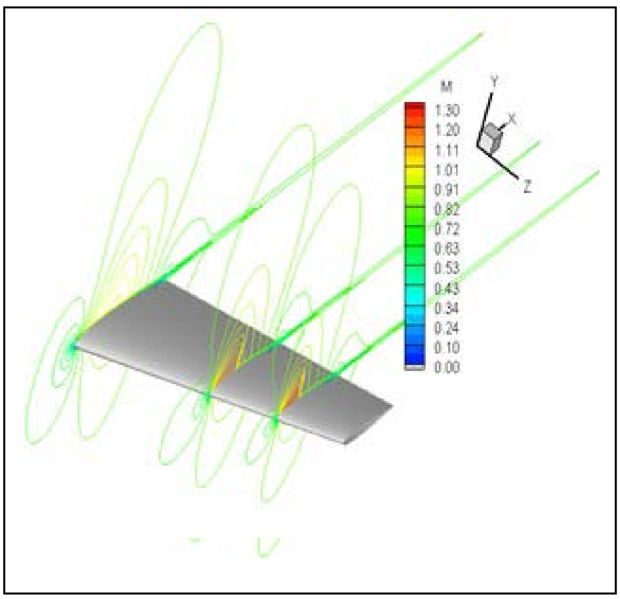



**Figure 10. Iso-Mach lignes for different stations along of span winge :for z/b=0.5 &z/b=0.5**

## 6. Conclusion

In this study, we simulate a transonic flow around a wing with symetrical airfoile symmetrical; among the conclusions reached:

The prediction of wave drag around airfoil and wing operating in the transonic mode and progression of shock waves on the airfoil surface depending on the Mach number, The appearance of the shock wave that interacts with the boundary layer significantly increases the drag and induced wave drag created at a divergence Mach [9,10], that promotes the separation of air streams Also lift loss and creation of aeroelasticity problems. Hence, the choice of airfoils for UAV transonic wing with their aerodynamic characteristics remain a challenge to raise.

## References

[1] GAO Chao, Luo Chijun, “Numerical solution of the unsteady Euler equations for airfoils using approximate boundary conditions”, Acta mecanica sinica, vol 5 N°19, oct 2003.

[2] A.Jamesson.”Numerical solution of the Euler equations by finite volume methods using Runge Kutta time stepping schemes.” Elsevier science publication co.,inc.,1983, p1-11.

[3] Bruno Goffert, “Euler Equations Applied To Flow Over Naca0012”, 20th International Congress of Mechanical Engineering November 15-20, 2009, Gramado, RS, Brazil.

[4] Dennis L. Huff, “Analysis of Viscous Transonic Flow Over Airfoil Sections”, NASA Technical Memorandum 88912, AIAA-87-0420, 25th Aerospace Sciences Meeting, Nevada,January 12-15, 1987.

[5] W.P., Jones, and B.E., Launder, The Prediction of laminarization with a two-equation model of turbulence, International. Journal of Heat and Mass Transfer, 15 (2), pp. 301- 314, 1972.

[6] D.C., Wilcox, Turbulence Modeling for CFD, 2nd ed., DCW Industries, La Canãda, CA, 1998.

[7] F.R., Menter, L.C., Rumesy, Assessment of two-equation turbulence models for transonic Flows, AIAA Paper 94-2343, June 1994.

[8] Transonic flow around an airfoil Transonic - Wikipedia, the free encyclopedia

[9] Novel Kumar Sahu, Mr.Shadab Imam, “Analysis of Transonic Flow over an Airfoil NACA0012 using CFD”, IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 4, April 2015. Pp.379-389.

[10] J. Fořt, J. Fürst, K. Kozel, A. Jirásek, M. Kladrubský, “Numerical Solution of 2D and 3D Transonic Flows Over an Airfoil and a Wing”, Fluid Mechanics and its Applications Volume 73, 2003, pp 211-216.