Heat and Technology
Vol. 32, n. 1-2, 2014

NUMERICAL INVESTIGATION OF UNSTEADY FLOW AROUND AN AIRFOIL

MADANI Y. H, BENSAADA. M and RACHED .D

Département de Génie Mécanique, Lab RTFM, ENPO, Oran ALGERIE

ABSTRACT

During this work, we simulated an unsteady flow around an airfoil type NACA0012 using the Fluent software. The objective is to control the code on the one hand and on the other hand the simulation of unsteady flows. By simulating an unsteady flow Reynolds number (Re = 6.85 * 106) and Mach number (M = 0.3), we have the flowing with a grid (mesh) adequate numerical results and experimental data are in good agreement. To represent the results of the simulation we have validated by comparing the values of aerodynamic coefficients with those of experimental data

1. INTRODUCTION

Predetermination of unsteady flow was always a concern for the research because the implementation difficult in laboratories and sophisticated equipment requirements. Our objective of this study is to control the simulation of unsteady flows around structures. Using the finite volume method and Spalart-Allmars model, designed for applications in Aerospace, our results were in good agreement with experimental data. By simulation studies predeterminations became very easy to prepare, this gain is the result of the development of computational methods and hardware remarkable revolution. So mastery of computers has become indispensable for such studies simulation. Representations available in the Fluent software allowed me to understand the discipline of an unsteady flow.

For this work we simulated unsteady flow around a NACA0012 profile type in the following conditions:
Iso-thermal wall at 300°k;
Fluid temperature 300°K;
Atmospheric pressure, P = 101325 Pa;
Reynolds number, Re = 6.84 .106;
Mach number,
M = 0.3;
Angle of incidence, $\alpha = 4$;
Relationship for the viscosity of Sutherland.

2. TURBULENCE MODELING

The numerical model used in this simulation is based on the Navier Stokes Equations for incompressible flow. The turbulence is resolved via the Reynolds Averaged approach, wich consist of resolving the averaged variables and the fluctuations. This is done by the well known Shear Stress Transport model of Menter group. The formulation can be found in the technical scientific literature.

3. GEOMETRY AND MESH:

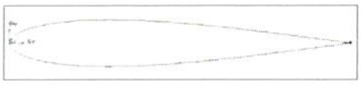

Figure 1 : Naca0012 profile.

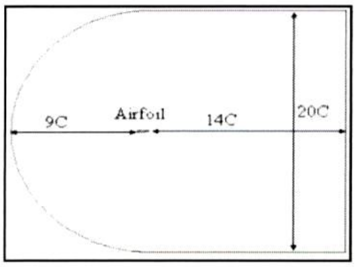

Figure 2 : Domain of camputation

Figure 3 : Mesh near the profile

4. STUDY OF DEPENDENCE IN MESH

In order to check the mesh dependency or not the simulation of steady flow at 10° incidence is made by calculating four cases by conducting a mesh adaptation to the walls for the 3rd and 4th cases, the results are affirmative for dependence and the thickness of the first mesh affects the simulation results.
Values of lift coefficient were improved after modifications of the mesh, however the accuracy of convergence is higher

for the first adaptation and stepped back for seconds. It is advisable to choose a good convergence criterion not to waste time calculating assets without a converged solution. Figures 4 to 8.

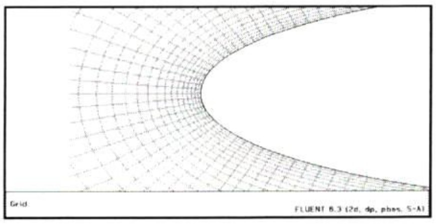

Figure 4 : Mesh whitout adaptation

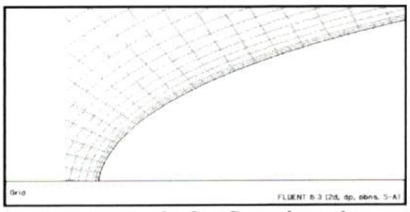

Figure 5, Mesh after fisrt adaptation

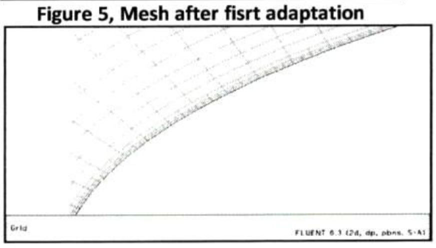

Figure 6 : Mesh after second adaptation

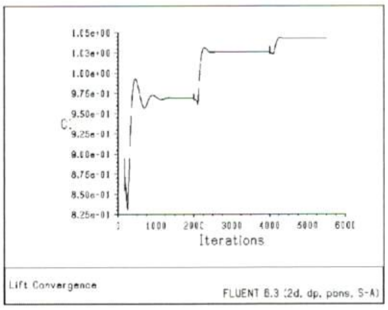

Figure 7 : hystorical convergence of lift

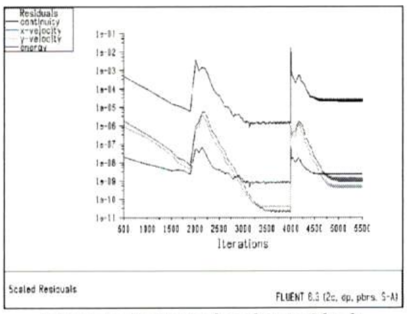

Figure 8 : hystorical scales residuals

5. VALIDATION OF RESULTS

To validate our results, we compared the values of the coefficients of drag and lift with the experimental data obtained by N. And Gregory N. P. Welby. These values are taken for both cases for each incidence angle. V1 is taken after the convergence of the continuity equation for a test 10^3 and V2 after the zero convergence of these coefficients. Figures 9 & 10.

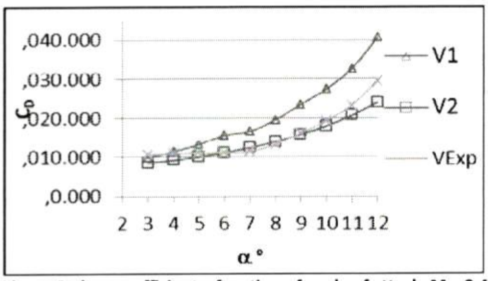

Figure 9: drag coefficient a function of angle of attack, M = 0.4

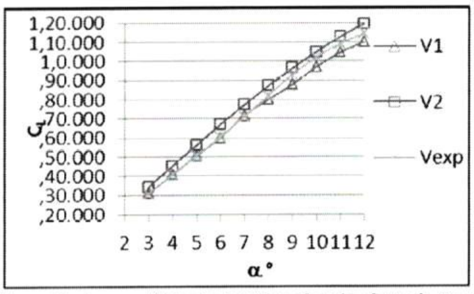

Figure 10 : drag coefficient a function of angle of attack, M = 0.5

6. REPRESENTATIONS OF STEADY RESULTS

We represented the stationary results in order to use them to initiate the unsteady calculation and avoid oscillations composition in the transient portion of the flow. Figures 11 to 14.

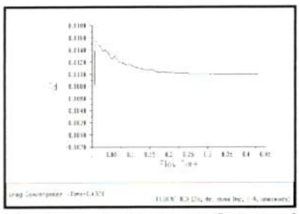

Figure 11 : Drag convergence for steady flow

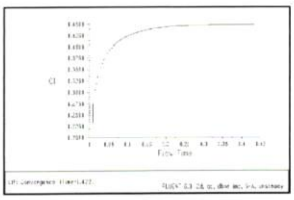

Figure 12 : Lift convergence for steady flow

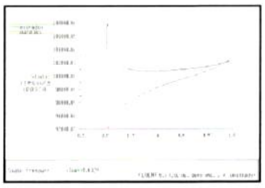

Figure 13 : Distribution of Static pressure around the airfoil

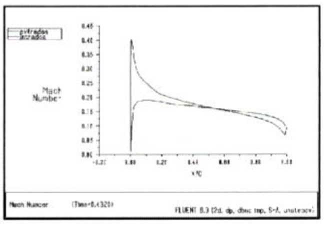

Figure 14 : Distribution of Mach number around the airfoil

7. REPRESENTATIONS OF UNSTEADY RESULTS:

According to [13] and {14], the simulation of an unsteady flow are done by use of a static pressure pulse to the input of area to get the Mach 0.32> M> 0.278. This pulse is defined by the general equation [$P = A \sin(\omega.t) + Patm$]. The figures above (15 to 18) highlight the difference between steady results and those unsteady.

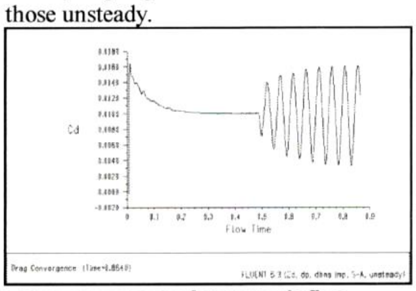

Figure 15: Drag convergence for unsteady flow

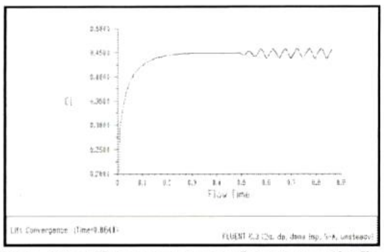

Figure 16 : Lift convergence for unsteady flow

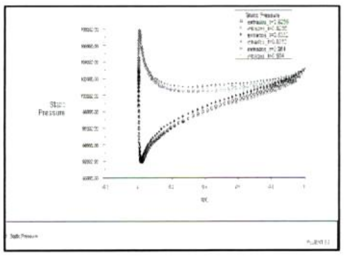

Figure 17: Distributions of instantaneous static pressure around the airfoil

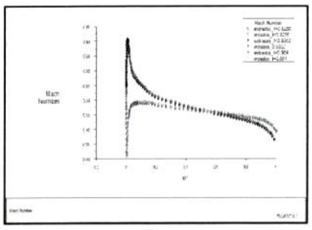

Figure 18 : Distributions of instantaneous static pressure around the airfoil

8. THE STATIC PRESSURE INSTANTANEOUS ISO-CONTOURS

The different Iso-contours are represented to highlighting the diffences between the contours of the instantaneous static pressure. Figures 19 to 24.

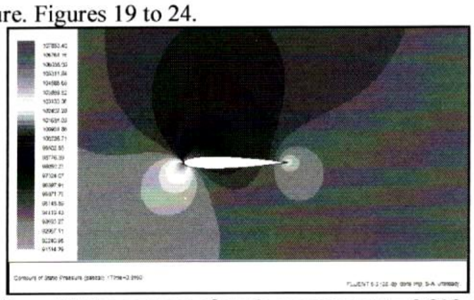

Figure 19: Iso-contour of static pressure at t = 0.816 s.

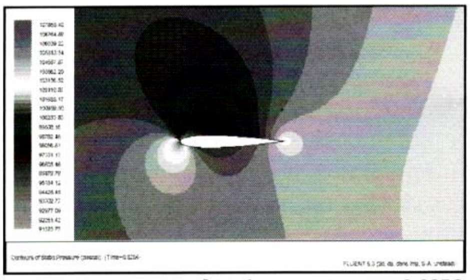

Figure 20: Iso-contour of static pressure at t = 0.8256 s.

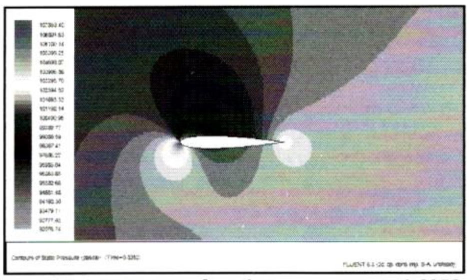

Figure 21: Iso-contour of static pressure at t = 0.8352 s.

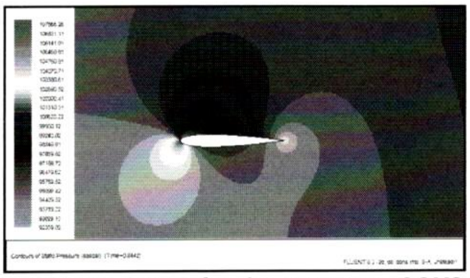

Figure 22: Iso-contour of static pressure at t = 0.8448 s.

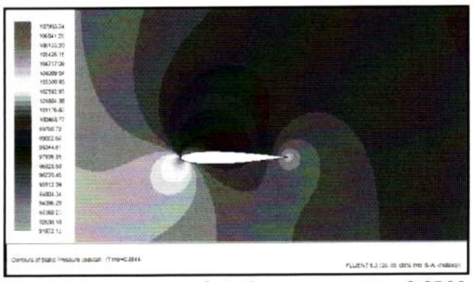

Figure 23: Iso-contour of static pressure at t = 0.8544 s.

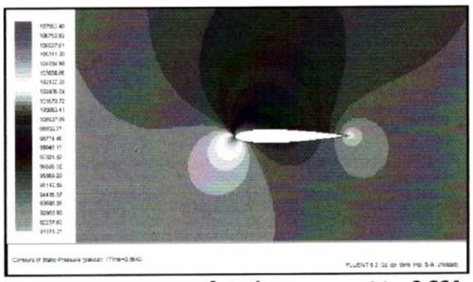

Figure 24: Iso-contour of static pressure at t = 0.864 s.

9. CONCLUSION:

The historical convergences of the aerodynamic coefficients highlight the influence of changing the speed at infinite upstream of these coefficients. The oscillation of coefficients produces a vibration of aircraft wings will, in view of the rigidity of the wing is the cause vibration by the flowing fluid.

The Iso-contours of the instantaneous static pressure represented illustrate the dependence of the flow time, so the flow is unsteady and the pressure distribution around the profile changes from one moment to another by oscillating the aerodynamic coefficients. Thus the oscillation of static pressure is defined boundary conditions at the origin of the variation of the speed and therefore Mach number. To conclude our study, with an adequate grid and well-chosen approach the numerical results and ones experimental are in good concordance. so boundary conditions must be well defined.

10. REFERENCES

[01] P. Spalart and S. Allmaras. A one-equation turbulence model for aerodynamic flows. Technical Report AIAA-92-0439, American Institute of Aeronautics and Astronautics, 1992.

[02] L. Chatellier. Course: unsteady fluid mechanics, 2005-2006.

[03] Dr. Robert C. Nelson, Flight Stability and Automatic Control, second edition. Department of Aerospace and Mechanical Engineering, University of Notre Dame. International Editions 1998.

[04] Documentation provided with Fluent software, Fluent 6.3 User's Guide 2006.

[05] D. Choudhury. Introduction to the Renormalization Group Method and Turbulence Modeling. Fluent Inc. Technical Memorandum TM-107, 1993.

[06] Dr. Rémi Bourguet, Physical analysis and modeling of unsteady compressible turbulent flows around airfoils by statistical approaches high fidelity and small size in the context of fluid-structure interaction. Issued by the Institut National Polytechnique de Toulouse Specialty: Fluid Dynamics, 2008.

[07] J. O. Hinze. Turbulence. McGraw-Hill Publishing Co., New York, 1975.

[08] B. E. Launder and D. B. Spalding. Lectures in Mathematical Models of Turbulence. Academic Press, London, England, 1972.

[09] T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu. A New k-ϵ Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation. Computers Fluids, 24(3):227{238, 1995.

[10] D. C. Wilcox. Turbulence Modeling for CFD. DCW Industries, Inc., La Canada, California, 1998.

[11] F. R. Menter. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA Journal, 32(8):1598{1605, August 1994.

[12] M. M. Gibson and B. E. Launder. Ground E_ects on Pressure Fluctuations in the Atmospheric Boundary Layer. J. Fluid Mech., 86:491{511, 1978

[13] Ali Triki, Ezzeddine Hadj-Taïeb, 'Numerical simulation for one dimensional open channel transient flow', International Journal of Heat and Technology, Volume 26 No 1 (2008).

[14] K. Jha Basant, M.L. Kaurangini, 'UNSTEADY COUETTE FLOW IN PARALLEL PLATES FILLED WITH POROUS MATERIAL WITH SUCTION/INJECTION', International Journal of Heat and Technology, Volume 29 No 1 (2011).