NUMERICAL INVESTIGATION OF UNSTEADY FLOW AROUND AN AIRFOIL

MADANI Y. H, BENSADA. M and RACHEDI. 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-Allmaras model, designed for applications in Aerospace, our results were in good agreement with experimental data. By simulation studies predetermination 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, which 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.](image1)

![Figure 2: Domain of computation](image2)

![Figure 3: Mesh near the profile](image3)

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.

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 10^5 and V2 after the zero convergence of these coefficients. Figures 9 & 10.

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.
8. THE STATIC PRESSURE INSTANTANEOUS ISO-CONTOURS

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

Figure 18: Distributions of instantaneous static pressure 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 $P = A \sin(\omega t \delta) + Patm$. The figures above (15 to 18) highlight the difference between steady results and those unsteady.

Figure 14: Distribution of Mach number around the airfoil

Figure 15: Drag convergence for unsteady flow

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

Figure 16: Lift convergence for unsteady flow

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

Figure 17: Distributions of instantaneous static pressure around the airfoil

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

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


