ISSN 1392 - 1207. MECHANIKA. 2011. 17(2): 193-196

Numerical simulation of a flow around an unmanned aerial vehicle

S.M.A. Meftah*, B. Imine, O. Imine***, L. Adjlout******

**Mechanical Engineering Faculty, USTO Oran, B.P 1505 El Mnaouer U.S.T, Oran, Algeria, E-mail: hmeftahdz@yahoo.fr*
***Mechanical Engineering Faculty, USTO Oran, B.P 1505 El Mnaouer U.S.T, Oran, Algeria, E-mail: imine_b@yahoo.fr*
****Mechanical Engineering Faculty, USTO Oran, B.P 1505 El Mnaouer U.S.T, Oran, Algeria, E-mail: imine_o@yahoo.fr*
*****Mechanical Engineering Faculty, USTO Oran, B.P 1505 El Mnaouer U.S.T, Oran, Algeria,*
E-mail: adjloutl@yahoo.fr

crossref http://dx.doi.org/10.5755/j01.mech.17.2.339

1. Introduction

The unmanned aerial vehicle (UAV) is a small plane which flies in an autonomous way with only one virtual pilot on board. That does not mean that the plane is completely out of control once it leaves the track. The presence of a human being in the loop of control is essential. But the operator is on a station on the ground, either on ground, or at sea or in a control center. He can intervene at any time, examine the situation and the rules of engagement, and stop the mission if necessary. The concept of an UAV resounds in several research laboratories of any developed country. The first instigators of this type of the projects were the soldiers. Indeed, the armies of many countries prefer to rather send UAV's for the recognition in the high-risk zones that man. This type of flying machine can take much various missions, extending from the control of the scientific experiments to the intelligence collecting the monitoring during the day or the night.

The numerical simulation in the field of aerodynamics is relatively recent research tools. The specialists raise the question of knowing if the field of aerodynamics is purely theoretical or numerical, they estimate however that it is theoretical, because of the many numerical tests necessary for the stage to insufficiencies of the knowledge of the calculation methods [1]. Since a score of years, one can say that numerical aerodynamics lost much of its empirical nature due to the convergent efforts of mathematics to solve the nonlinear problems involved in the calculation methods in aerodynamics and also thanks to the invention of the very fast machines of calculation and the development of data processing.

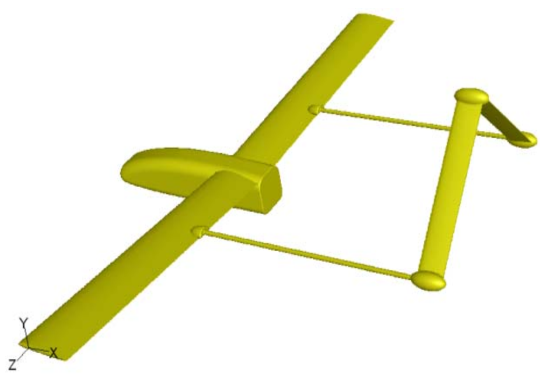

Fig. 1 The solid model of the UAV

In this present work, a numerical simulation of a flow around an UAV is presented. The first phase of this work consists in using plans of construction of this UAV to create a solid model which specifies its external geometry in Solid Works (Fig. 1). This solid model is then exported in to a programme of the grid generation in order to return its geometry adapted to aerodynamic calculations and tested in various flying conditions [2]. A computer code of computational fluid dynamics (CFD) [3] is used to obtain estimates of the coefficients of lift and drag to 20 meters per second. The model of turbulence Spalart-Allmaras [4] is used to leap the transport equations of Navier-Stokes in order to correctly predict the complex flow around the plane. A study of the independence of the grid is then carried out to determine the precision of the grid of calculation used for analysis CFD.

2. Description of the UAV

This UAV as shown in Fig. 2 is 2 meters long and provided with a pendular wing of the scale of 2.9 m. the profile of this wing is of type Clark yh with a cord of 0.236 m, stalled on the fuselage of 4° with a dihedral of 4°. The empennage group of the shape of V reversed is profiled with Naca 0012 and attached to the wings by two beams which give also the name of an UAV to twin-boom. These UAV's aspire of a remarkable stability to the no desirable movements of roll. This UAV is designed to have a pusher engine which will be embarked behind the fuselage.

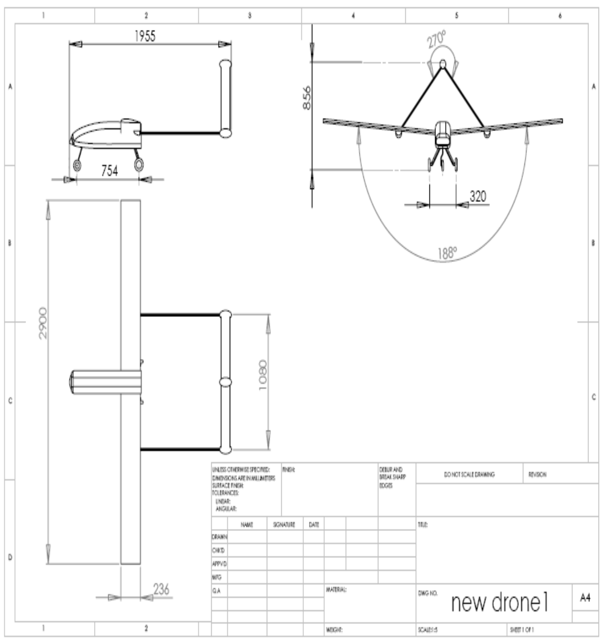

Fig. 2 Description of this UAV

3. Transport equations and turbulence model

The flow around the plane is considered turbulent asymmetrical. The general forms of the transport equations can be written in Cartesian coordinates [5] as:

- continuity equation

$$\frac{\partial}{\partial x_j}\left(\bar{\rho}\,\tilde{U}_j\right)=0 \qquad (1)$$

- momentum conservation equation

$$\frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{U}_i\tilde{U}_j\right)=\bar{\rho}g_i-\frac{\partial \bar{p}}{\partial x_i}-\frac{\partial}{\partial x_j}\left(\rho u_i'' u_j''\right)+$$
$$+\frac{\partial}{\partial x_j}\left[\bar{\mu}\left(\frac{\partial \tilde{U}_i}{\partial x_j}+\frac{\partial \tilde{U}_j}{\partial x_i}\right)-\frac{2}{3}\bar{\mu}\frac{\partial \tilde{U}_k}{\partial x_k}\delta_{ij}\right] \qquad (2)$$

- turbulence model

The model of turbulence used in this present work is a model with a transport equation for the v quantity suggested by Spalart and Allmaras [4]

$$\frac{\partial\left(\bar{\rho}\tilde{v}\right)}{\partial_t}+\vec{\nabla}\left(\bar{\rho}\tilde{\vec{V}}\tilde{v}\right)=\vec{\nabla}\left[\frac{\bar{\mu}+\bar{\rho}\tilde{v}}{\partial SA}\vec{\nabla}\tilde{v}\right]+P_v-D_v \qquad (3)$$

On the terms of production and destruction are defined as follows

$$P_v=C_{b1}C_{\mu}\tilde{S}\,\bar{\rho}\tilde{v}+C_{b1}\frac{\bar{\rho}\left(\vec{\nabla}\tilde{v}\right)^2}{\partial_{SA}} \qquad (4)$$

$$D_v=C'_{\omega_1}f_{\omega}\frac{\bar{\rho}\left(\vec{\nabla}\tilde{v}\right)^2}{\rho d^2} \qquad (5)$$

with

$$\left.\begin{aligned} f_x &= g\left(\frac{1+C_{\omega 3}^6}{g+C_{\omega 3}^6}\right)^{\frac{1}{6}} \\ g &= r+C_{\omega 2}\left(r^2-r\right)\end{aligned}\right\} \qquad (6)$$

and

$$\left.\begin{aligned} M_t &= \bar{\rho}\tilde{v}\frac{\chi^3}{\chi^3+C_{v1}} \\ \chi &= \frac{\bar{\rho}\tilde{v}}{\mu}\end{aligned}\right\} \qquad (7)$$

and the coefficients of closing are given by the following values:

$$\left.\begin{aligned} &\sigma_{SA}=\frac{2}{3};\quad C_{b1}=0.1355;\quad C_{b2}=0.622;\\ &C_{v1}=7.1;\quad C_{\omega 1}=0.3;\quad C_{\omega 3}=2.\end{aligned}\right\} \qquad (8)$$

4. Results

Several compilations were carried out for various angles of incidence to determine the lift and drag. Each case was carried out at the same speed: 20 m/s. The angle of attack extended from 3 to 18°.

The evolution of the drag coefficient according to the incidence is presented in Fig. 3. It is a parabolic type tendency similar to those found on the planes in general. In particular, one notes the increase in the drag with the increase in the angle of incidence. This curve is presented with drag of the isolated wing which equips the UAV in question and which was obtained by wind tunnel test. The two curves follow the same tendency separately which is normal because variation of the drag between the two curves corresponds to the drag of the remainder of the UAV which varies very slightly with the incidence.

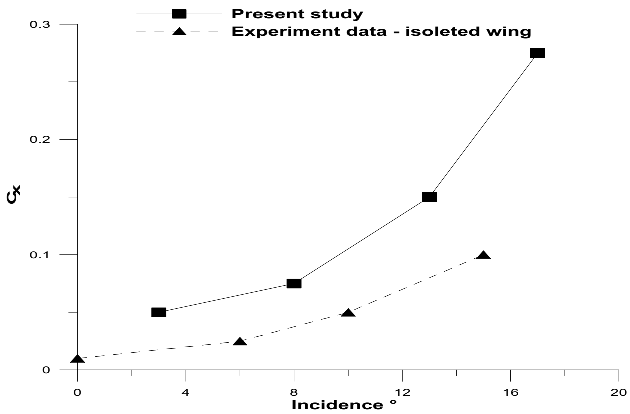

Fig. 3 Evolution of the drag coefficient

Fig. 4 treats the evolution of the lift according to the incidence for the UAV and the isolated wing. An agreement of the two curves is noted in the linear zone which is explained by the fact what it is the wing which contributes in a great proportion to the total lift produced on the plane. The other parts of the plane, in particular the stabilizer; contribute slightly to weak and moderate incidence. However the lift of the isolated wing exceeds that of the plane for the bigger angles of incidence. This is dues with the fact that the Reynolds number of the tests of the isolated wing is much higher than that of the plane tested numerically which leads to higher maximum *Cz* for the

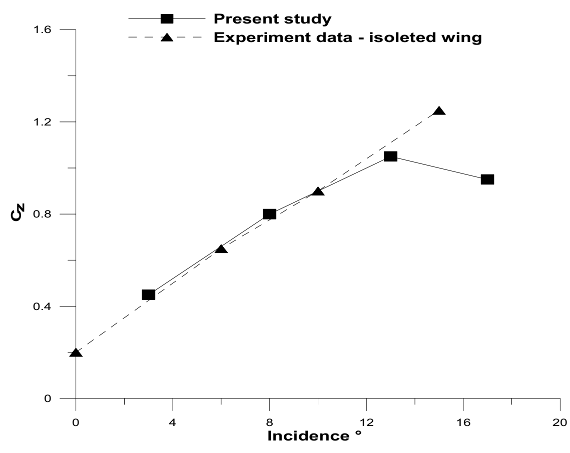

Fig. 4 Evolution of the lift coefficient

isolated wing. Numerical calculations show well the phenomenon of unhooking of the UAV which is located at approximately 12°. It should be noted that this incidence which appears is weak only seemingly because it should be mentioned that the reference of measurement of the angle of attack is the base of the fuselage and that the wing is fixed with 4°.

The map of shear stress along X is shown in Figs. 5 and 6 for respectively an incidence of 0° and 15°. A flow return is noted on almost the totality of the wing, in particular, it is much accentuated in the central part. This effect is expected because the wing is not twisted and the great deflections of the air are perceptible in this area. The separation of the flow recorded on the nose of the fuselage still persists with this angle of incidence.

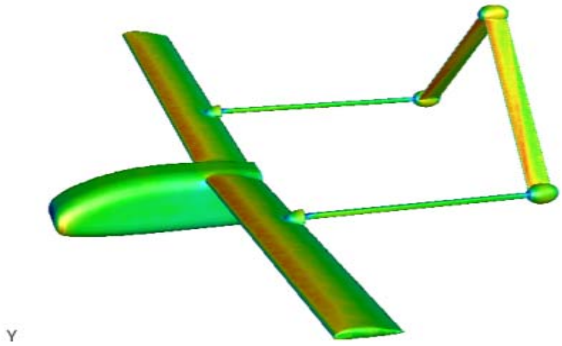

Fig. 5 Chart of shear stress for i = 0°

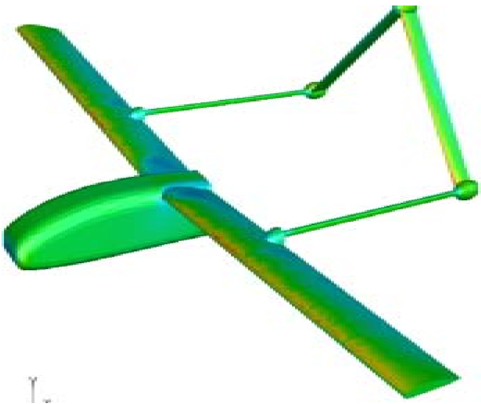

Fig. 6 Chart of shear stress for i = 10°

Fig. 7 shows the chart of the parietal flow recorded on the level of the wing root with the fuselage with 15° of incidence. It is a representation of the streamlines of the flow inside the internal zone of the boundary layer around the UAV which is completely turbulent. It is a very useful chart for the draftsman of the aerodynamic forms because it reveals the design defects characterized by the appearance of the zones of flow separated colour blue as shown in the Fig. 8 on the wing and on the level from the bases from fastener from the beams.

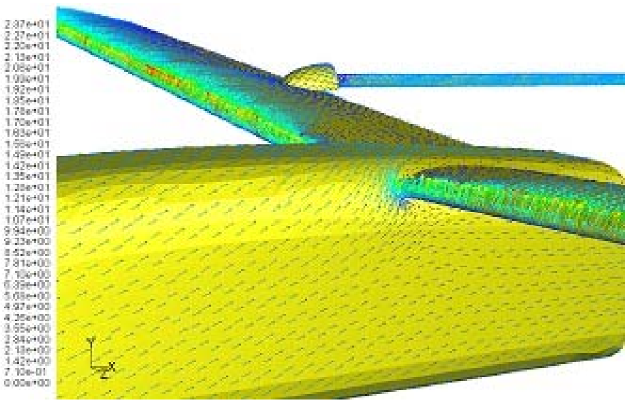

Fig. 7 Threads of current in the vicinity of the root

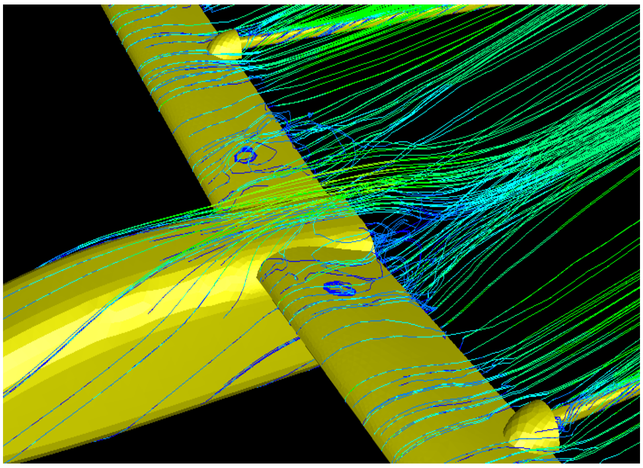

Fig. 8 Visualization of the zones of separation for i = 15°

Fig. 9 shows the two marginal swirls which release from salmons of the two wings of the UAV to the passage of the flow for the incidence 15°, the size of the marginal swirl for such an incidence is considered to be considerable. Moreover, one notes the complexity of the flow around the plane for an incidence judged like the beginning of unhooking.

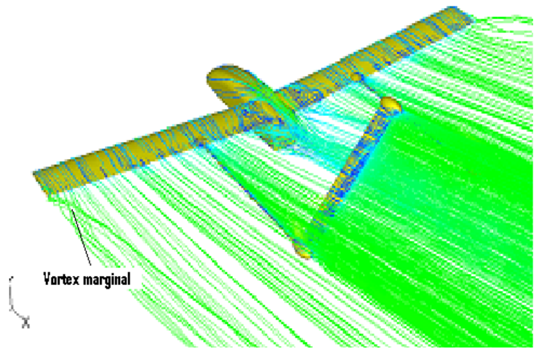

Fig. 9 Visualization of the tip vortex

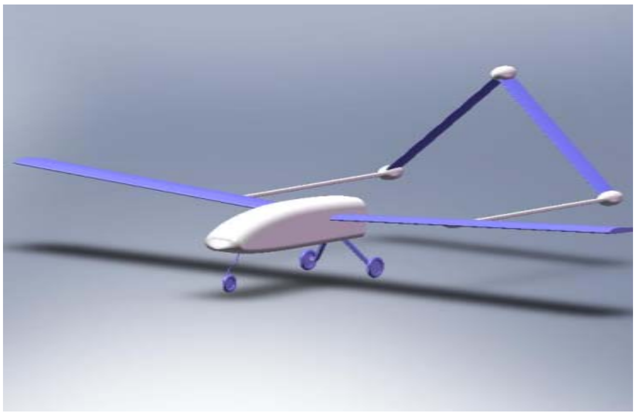

Fig. 10 Correcte UAV

5. Conclusion

The present study was undertaken to observe the aerodynamic performances of an UAV by using a CFD computer code. This concept of design of flying machines is judged like an alternative to the wind tunnel tests for a possible reduction of the design cost. As shown in Fig. 10, this work brought moreover, certain important aerodynamic corrections around this UAV such as the aerodynamic careenage of the nose of the fuselage and the setting of the bases of fastener of the beams on the level of the under-surface of the wings

References

1. **Ronzheimer, A.** 2005. Post-parameterization of CAD-geometries using freeform deformation and grid generation techniques, Notes on Numerical Fluid Mechanics and Multidisciplinary Design, vol.87: 382-389.
2. **Flores, J.; Reznick, S.G.; Hol, T.; Gundy, K.** 1988. Transonic Navier-Stokes solutions for a fighter-like configuration, Journal of Aircraft 25, October 1988: 875-881.
3. **Bhaskaran, R.** FLUENT Short Course, http://instruct1.cit.cornell.edu/courses/fluent. Includes step-by-step tutorials and problem sets. Chapters: Introduction to Computational Fluid Dynamics; Laminar pipe flow; Turbulent pipe flow; Compressible nozzle flow; Airfoil flow.
4. **Li, J.; Li, F.; Qin, E**. 2000. Numerical simulation of transonic flow over wing-mounted twin-engine transport aircraft, Journal of Aircraft, vol.37, No.3: 469-478.
5. **Benhamza, M.E.; Belaid, F.** 2009. Computation of turbulent channel flow with variable spacing riblets, Mechanika 5(79): 36-41.

S.M.A. Meftah, B. Imine, O. Imine, L. Adjlout

BEPILOČIŲ LĖKTUVŲ SKRAIDYMO SKAITMENINIS MODELIAVIMAS

R e z i u m ė

Šiame darbe pateikiamas bepilotinių lėktuvų skraidymo skaitmeninis modeliavimas. CFD metodas taikomas kilimo ir stabdymo koeficientams nustatyti esant 20 m/s greičiui ir įvairiems atakos kampams. Kompleksiškai tiriant bepiločio lėktuvo skraidymą buvo panaudotas Spalarto ir Allmaraso turbulentinis modelis. Tyrimo rezultatai parodė, kad apskritai srauto struktūra gerai sutampa su nustatyta eksperimentiškai.

S.M.A. Meftah, B. Imine, O. Imine, L. Adjlout

NUMERICAL SIMULATION OF A FLOW AROUND AN UNMANNED AERIAL VEHICLE

S u m m a r y

In this present work, a numerical simulation on a flow around an UAV is presented. The CFD is used to obtain evaluations on the coefficients of lift and drag with flow velocity of 20 meters per second for various angles of incidence. The model of Spalart-Allmaras turbulence is used for the investigation of the complex flow around the UAV. The results of the CFD indicate that the complex structure of the flow is well compared with reality.

С.М.А. Мефтах, Б. Имине, О. Имине, Л. Адджлоут

ЧИСЛЕННОЕ МОДЕЛИРОВАНИЕ ПОЛЕТА БЕСПИЛОТНЫХ САМОЛЕТОВ

Р е з ю м е

В данной работе представлено численное моделирование полета беспилотных самолетов. Для оценки коэффициентов подъема и сопротивления при скорости полета 20 метров в секунду и различных углах атаки использован CFD метод. При комплексном исследовании полета беспилотного самолета была использована турбулентная модель Спаларта-Аллмараса. Результаты исследования показали, что, в общем, структура потока хорошо соответствует эксперименту.

Received June 02, 2010
Accepted April 05, 2011