THE IMPACT OF DIFFERENT TURBULENCE MODELS AT ANSYS FLUENT OVER THE AERODYNAMIC CHARACTERISTICS OF ULTRA-LIGHT WING AIRFOIL NACA 2412

THE IMPACT OF DIFFERENT TURBULENCE MODELS AT ANSYS FLUENT OVER THE AERODYNAMIC CHARACTERISTICS OF ULTRA-LIGHT WING AIRFOIL NACA 2412

Cvetelina VELKOVA [1]
ts.velkova@naval-acad.bg

Terence BRANGER [2]
terence.branger@orenge.fr

Francesco MOTTA CALDERON [2]
calderon.francesco@gmail.com

Camille SOULIER [2]
camille.5@hotmail.fr

1: eng. PhD at the Department of Technical Mechanics, Naval Academy Nikola Vaptsarov, BULGARIA
2: Generalist engineering student at the ECAM Strasbourg-Europe, FRANCE

Abstract:

The purpose is to assess who would be the adequate turbulence models' at ANSYS Fluent at the calculation of the aerodynamic characteristics of ultra-light aircraft wing airfoil NACA 2412. A numerical study was performed on an NACA 2412 airfoil typical for Cessna 172 Sky hawk to explore the aerodynamic coefficients at Reynolds number of 3x106. The choice of optimal turbulence model used for NACA 2412 airfoil is made by comparison of its aerodynamics characteristics calculated using three different turbulence models' at ANSYS Fluent environment. All calculations were made using Spalart-Allmaras, k-ω SST, and– DES (Detached-Eddy Simulation) k-ω SST turbulence models' with CFD code Fluent. Relying on the experience gained in this field it can be concluded that used turbulence models' gave relatively good results for aerodynamic characteristics of 2D wing airfoil NACA 2412. But based on this computational study it can be concluded that DES k-ω SST is most accurate in solving of this kind of problems.

Key words
Aerodynamics, ANSYS Fluent, wing airfoil, turbulence models, aerodynamic coefficients.

1. Introduction:

As lift to drag ratio give us an idea of the aerodynamic efficiency of an airplane, it is important to study it. A high L/D ratio means that the lift force overcomes the weight force so that the aircraft can fly over a long distance. [1]

Some information can be found on the "NACA 2412" profile like the lift and drag coefficient in function of the angle of attack on the web. For a Reynolds number of 1.10^{+6} the website gives us: [2]

On those graphs, if we consider only positive angles, we can see that the lift coefficient turning down after an angle of approximately 16 degrees. This phenomenon gives us the stall angle of the airfoil. In our experimentation we expect to have this same point at this same angle.

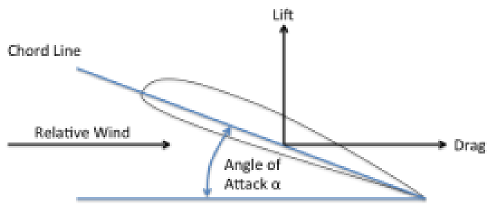

Fig. 1Aerodynamic forces on an Airfoil (Image from: http://code7700.com/angle_of_attack.html)

Previous study of the NACA 2412 airfoil [3] have shown that pressure is lower on the upper surface and reaches its maximum at the point of attack, but also that the flow velocity on the upper surface is faster than the lower surface of airfoil. It was also prove that lift force is larger about 22.5 times than drag force, which allows lifting the weight of the flying objects.

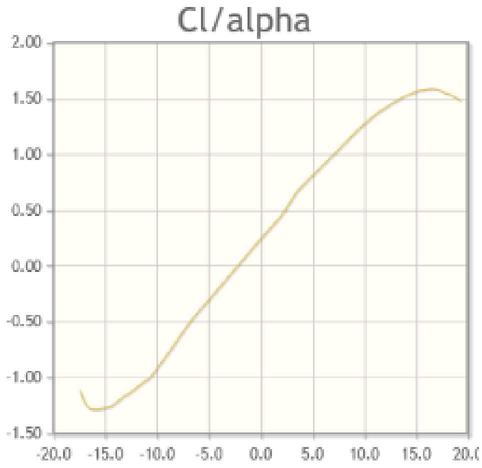

Fig. 2 Theoretical Lift coefficient as function of the angle of attack

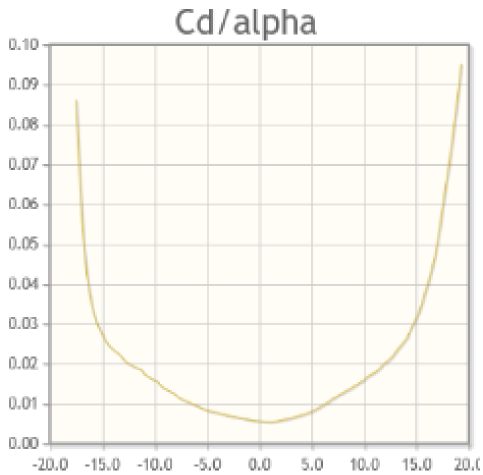

Fig. 3Theoretical Drag coefficient as function of the angle of attack

In this study, as we said before, the purpose is to assess which would be the optimal turbulence models.

So in order to realize it, the calculation of 'Spalart-Allmaras, k-ω SST, k-ε and– DES (Detached-Eddy Simulation) k-ω SST turbulence models' has been realized using CFD code FLUENT 2015 solver due to its advantages in diverse blocking techniques to create structured meshes.

2. Geometry and Mesh:

Process of airfoil design :

Coordinates of NACA 2412 are taken from an online airfoil database.

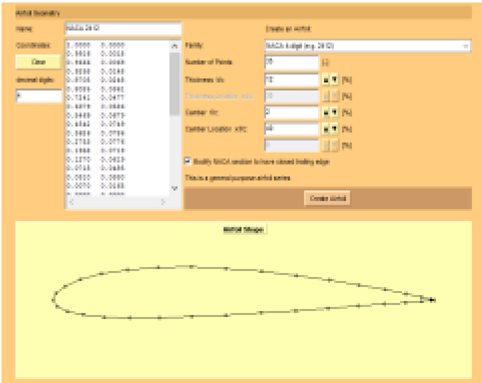

Fig. 4 Basic airfoil profile of NACA 2412, JAVAFOIL [4]

However, the following one hasn't a closed structure so in order to enhance it we use the function "trailing edge gap" of JAVAFOIL software and then we export the 35 coordinates for upper and lower surface to obtain a smooth profile. Figure 1 shows the basic airfoil profile of NACA 2412. JAVAFOIL is the analysis software, which gives us analysis data. Those who interest us are the coefficient of lift and drag in function of the different angle of attacks, [4]. Therefore, NACA 2412 airfoil has a camber of 2% located 40% back from the leading edge (0.4 c), with a maximum thickness of 12%.

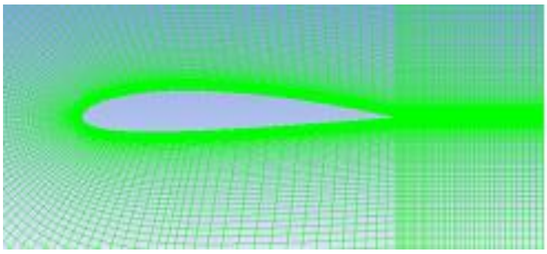

Fig.5 NACA 2412 profile

Grid generation :

The most appropriate type of mesh, and the one has been using to simulate flow around the airfoil is the C-mesh type, which in its simplest form consists of a semicircle with a radius of 5m in the area where flow comes with center point at the trailing edge acting as the inlet and a rectangle with a surface of 50m² in the area of flow exit acting as the outlet as shown in Figure 2.1 and 2.2. In order to be more specific in our calculations, we have split our flow domain into six sections as seen below. [5]

The mesh density i.e., number of nodes at the upper and lower section of the airfoil is greater in order to provide a better visualization of flow field near the surfaces, where is possible acceleration of the flow.

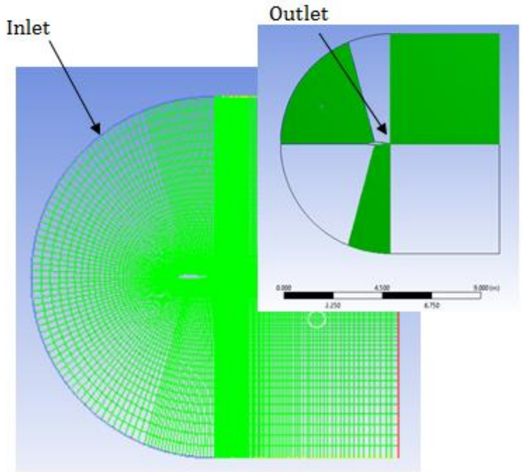

Fig. 6 Zoom on NACA 2412 airfoil meshing

The grid is a coarse quad map structure mesh and it has 30 000 cells and 30 300 nodes. For all the calculation, the grid is strongly compressed close to the wing airfoil contour, and particularly around the upper and lower culmination point.

Boundary conditions and initial values :

Velocity inlet, symmetry, wall and pressure outlet are the boundary types used for the simulations in this study. Those boundary conditions specify the flow variables on the boundaries of the computational domain. Other parameters vary depending on the turbulence model being implemented which in this case is the Spalart-Allmaras, k-ω SST, k-ε and– DES (Detached-Eddy Simulation) k-ω SST turbulence models.
In relation to the atmosphere where it will perform simulation it has decided to locate it at sea level. So we are talking about atmospheric conditions of density 1.225 $kg.m^{-3}$, pressure P = 101325 Pa and viscosity of 1.71*10-5 $Pa.s^{-1}$. [5]
The inlet velocity and the internal field of computational domain are of fixed value with a magnitude of 43.81 m/s which corresponds to a Reynolds number of 3.11 x 10^6 at 280 K. Knowing the Re number we can determine the Mach number which is the ratio between the inlet velocity and the speed of sound in the medium (340 m/s). Which gives us M = 0.128 < 0.8 meaning that we are in a subsonic regime. This coincides with the geometry of our wing airfoil put on the CESSNA 172 Skyhawk. [6]

Fluent :

The method of solution is iterative. The fluid CFD 2D wing airfoil model for the different angle of attack clarifies the unsteady flow field around the whole lift configuration by calculating the aerodynamic characteristic of wing airfoil.
Time step $\Delta t = 1 \times 10^{-3}$ is used, i.e., over 10 000 number of iterations (~1.0 s). In the Solver parameters box is selected the type Pressure-Based, as it is more indicated for low speed where we have incompressible flow or at least with very low compressibility factor. The solution is transient process because of the unsteady flow field. [5]

3. Simulation with different models:

ANSYS simulations' results give us access to the forces applied on both axes: Fx and Fy. It leads us to drag and lift forces with the following expression:

$$D = F_x.\cos(\alpha) + F_y.\sin(\alpha) \quad (1)$$

$$L = F_y.\cos(\alpha) - F_x.\sin(\alpha) \quad (2)$$

- α is the angle of attack in Radian

Moreover, lift and drag forces apply on the airfoils change in function of the angle of attack α. Both of them give us coefficients that we will calculate with mathematical relations:

$$C_D = 2D/\rho v^2 S \quad (3)$$

$$C_L = 2L/\rho v^2 S \quad (4)$$

- D is the drag force in N
- L is the lift force in N
- ρ is the air density in $kg.m^{-3}$
- v is the relative air velocity on the airfoil in $m.s^{-1}$
- S is the surface in m^2

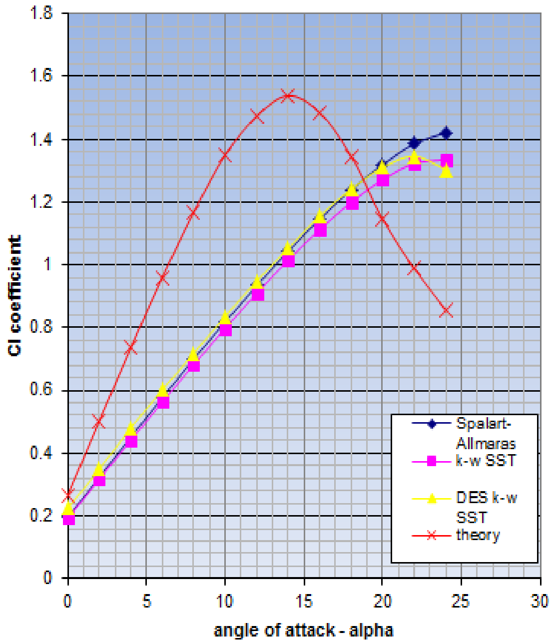

Fig. 7 Comparison between lift coefficient c_L of wing airfoil NACA 2412 with Spalart-Allmaras, k-w SST, DES k-w SST turbulence models and experiment (theory)

Spalart-Allmaras:

In the Spalart-Allmaras model we obtain a one-equation model that solves a modeled transport equation for the kinematic eddy turbulent viscosity. The Spalart-Allmaras model is supposed to gives us good results for our type of simulation. This model is a low-Reynolds number model, requiring the viscosity-affected region of the boundary layer to be properly resolved. In analyze of these two graphics we can say that comparatively to the NACA 2412 graphics, some problems are obvious. Indeed, concerning drag coefficients, the theoretical one approximate the real shape even if it don't increase correctly to reach a value near 0.1 linearly. The experimental is distorted too but not far of the NACA 2412 graphic on the shape of the curve point of view even if an extreme value (0.38) is way too huge.

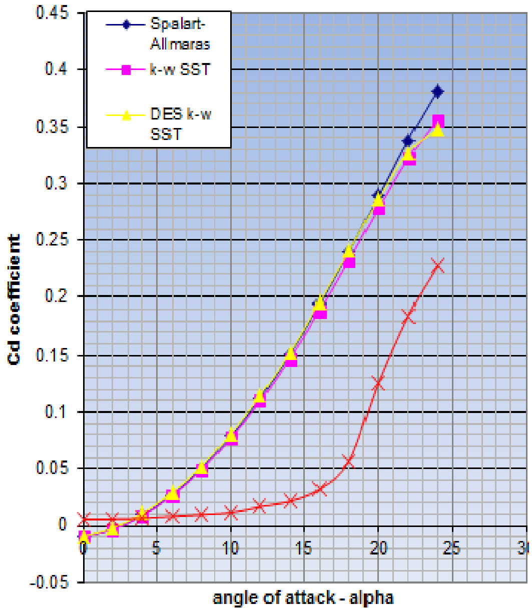

Fig. 8 Comparison between drag coefficient c_D of wing airfoil NACA 2412 with Spalart-Allmaras, k-w SST, DES k-w SST turbulence models and experiment (theory)

k-w SST :

The k-ω model is a two equation turbulence model, k represents the energy in the turbulence whereas ω represents the scale of the turbulence. On the other hand SST model, meaning Shear Stress Transport, combines k-ω model for the inner region (near the wall) and k-ε model for the free shear flow (far from the wall) where ε represents the dissipation of the turbulence energy.

Even if SST k-ω model produce too much turbulence levels in regions with large normal strain (stagnation regions or with strong acceleration) it's still a very used model. [7]

As we can see the result obtain is not as similar to the theory as we can hope. Especially for the Cl coefficient where we observe that between 20° and 22° the curve start going down, which means that according to this turbulence model the stall angle is between 20° and 22° or this airfoil is suppose to have a 14° stall angle. Therefore, in order to study properly the NACA 2412 airfoil, we should use another turbulence model even if the Cd curve is pretty correct and that the lift to drag ratio is respected.

DES k-w SST :

Various turbulent models and mesh procedures exist and have successfully been used. Referring to the large number of investigation in that area made until this days, we could forecast that DES k-ω SST (detached eddy simulation k-ω shear stress transport) turbulent model - which combines LES (Large Eddy Simulation) and RANS (Reynolds – Average Navier-Stokes) approaches – is quite suitable because of its advantages in preserving the efficiency and the accuracy within reasonable computer-time demands. Furthermore, DES k-ω SST produces accurate results for separated, high Reynolds external aerodynamics flow applications. [8]

The obtained numerical results for aerodynamic characteristics of Cl and Cd are approximated well with experimental data because it gives us a good trend of curves.

The results show us a stall angle estimated at 22° while the theory is about 14°.

However, it gives us approximately the same coefficient Lift meaning that relative error is quite suitable (ε = 12% by default). (Fig.1).

On the other hand, Drag coefficient curve obtained by the modelization looks to have the same trend until the 15° and then a gap appear provoking considerable differences between values. (relative error at stall angle : ε = 33%) (Fig.2).

There cannot be given the exact explanation why the error is so big reflected on the percentages. The cause can be because of the not sufficient mesh density, inappropriate selected size for the domain, the chosen value for the time step into the iteration process, the number of the iterations and etc. That why there is need the study to be expanded and for example to calculate the exact value of the time step taking into consideration the velocity magnitude.

4. Conclusion

We cannot say which of the three used turbulence models give us the best results onto this stage of our study. Because there is need the performed numerical study to be extended.

The main goal of this study is to propose an universal numerical approach that allows by the given geometrical model of the object, as in this case a wing profile to be calculated its aerodynamic characteristics. The suggested approach should be sufficiently accurate and efficient in order to improve the aerodynamics of the wings for this kind of ultra-light aircrafts.

To fulfill the purpose of that study, and hence to improve the aerodynamics of ulta-light aircraft wing, our task in analyzing the obtained results was to define which of those 3 model was the most akin to the theory. And finally we can confirm that the accuracy of the applied numerical DES k-ω SST model is effective but with some inaccuracy.

Thus, we suppose that our imprecision of 57% for stall angle are due to our 2D model that isn't representative of the real environment in which the airfoil is immersed.

Secondly, we do not have taken into account the wing airfoil's material (in aluminum).

Last but not the least; we have considered that the wind speed was uniform, and that is inappropriate.

5. Recommendations

How we can improve the developed by us numerical approach?

There are two general ways to improve this kind of numerical models:

1) By expanding the size of the computational domain. But for our case it will not help, because the problem we are solving for the flow calculations is 2D two-dimensional and the object around we are calculated the flow field is with simple geometry-wing airfoil.

2) Another way is by increasing the mesh density i.e make the mesh more coarse. We have to extend the study to check the last mentioned way.

Also to clarify more our study it should be repeated with the same models of turbulence, but with different types of the mesh i.e. also types of grid domain. In addition to C-type mesh, for example our study for the aerodynamics of wing airfoil NACA 2412 can be repeated but with O-type mesh.

Another aspect in which the study can be extended it is to perform it for other types of airfoils used to design the wings typical for other types of ultra-light aircrafts.

References:

[1] http://wright.nasa.gov/airplane/ldrat.html

[2]http://airfoiltools.com/airfoil/details?airfoil=naca2412-il

[3] VNU Journal of Science: Mathematics – Physics, Vol. 31, No. 2 (2015) 68-75

[4] Design of NACA 2412 and its Analysis at Different Angle of Attacks, Reynolds Numbers, and a wind tunnel test - Er. Shivam Saxena, Mr. Rahul Kumar

[5] BACHELOR THESIS TITLE: Aerodynamic analysis of the lift reducer device by flow transfer - Pau Font Prats

[6] Computational Investigation of Active Flow Control Parameters on the Aerodynamic Performance of NACA 2412 Airfoil - Bhanu Prakash Reddy Samala

[7] Rajesh Alayil · Indian Institute of Technology Madras

[8] Recent advances on the numerical modelling of turbulent flows - C.D. Argyropoulos , C. Markatos

ВЛИЯНИЕ НА РАЗЛИЧНИТЕ МОДЕЛИ НА ТУРБУЛЕНТНОСТ В СРЕДА ANSYS Fluent ВЪРХУ АЕРОДИНАМИЧНИТЕ ХАРАКТЕРИСТИКИ (АДХ) НА КРИЛЕН ПРОФИЛ NACA 2412 НА УЛТРА ЛЕК САМОЛЕТ

Цветелина Велкова
Катедра Техническа Механика, ВВМУ „Н.Й. Вапцаров“
ts.velkova@naval-acad.bg

Terence BRANGER
ECAM, Страсбург, Франция
terence.branger@orenge.fr

Francesco MOTTA CALDERON
ECAM, Страсбург, Франция
calderon.francesco@gmail.com

Camille SOULIER
ECAM, Страсбург, Франция
camille.5@hotmail.fr

Резюме

Предвид факта, че отношението подемна сила L към челно съпротивление D ни дава представа за аеродинамичната ефективност на самолета е важно тя да бъде изучена. Целта на настоящето изследване е да се прецени, кой може да бъде най-подходящият модел на турбулентност в среда ANSYS Fluent при изчисляване на аеродинамичните характеристики (АДХ)на крилен профил NACA 2412 на ултра лек самолет. Численото изследване е изпълнено за профил NACA 2412 типичен за самолет Cessna 172 Skyhawk за изследване на АДХ при числа на Рейнолд $Re=3x10^6$. Изборът на най-подходящ турбулентен модел за профил NACA 2412 е направен чрез сравнение на неговите АДХ получени с използването на три модела на турбулентност в среда ANSYS Fluent. Изчисленията са направени използвайки, Spalart-Allmaras, k-ω SST, и DES (Detached-Eddy Simulation) k-ω SST модели на турбулентност с помощта на CFD кода Fluent 2015 предвид неговото предимство в използването на разнообразни техники за изграждане на структурирани мрежи. Около профила е генерирана C – тип мрежа, с вход: полуокръжност с радиус 5 m на изхода: правоъгълник с площ $50m^2$. Флуидният обем е разделен на шест отделни повърхнини за по-точни изчисления на характеристиките на полето на течението. Мрежата е структурирана (coarse quad map) състояща се от правоъгълни елементи. Зададените гранични условия са: на входа е зададена скорост 43.81 m/s, при $Re=3x10^6$, профила е зададен като стена за да бъде изпълнено условието за непротекаемост на флуида, а на изхода е зададено налягане. Зададена входна скорост отговаря на число на Мах M = 0.128, т.е. дозвуков диапазон, което съвпада с геометрията на крилния профил използвана при самолет Cessna 172 Skyhawk, [6]. Процесът на решение е итеративен. Изграденият числен 2D модел на крилен профил NACA 2412 е реализиран за диапазон от ъгли на атака с цел изясняване на неустановеното поле на течението около цялата излетна конфигурация и пресмятане на АДХ на профила. Резултатите получени с реализирана числена симулация са за силите F_x и F_y като чрез формулите (1), (2), (3) и (4) са получени стойностите за подемна сила L към челно съпротивление D, и коефициентите им c_L и c_D.Използваният турбулентен модел Spalart-Allmaras е с ниска степен на точност и е характерен за ниски стойности на Re. От Фиг. 7 и Фиг. 8 се наблюдава, че резултатите получени с този турбулентен модел не са достатъчно точни. Използван е също така k-ω SST модел на турбулентност, който е с по-висока степен на точност и решава две преносни уравнения на потока. Но резултатите, получени и с този турбулентен модел не са достатъчно близки до приетите експериментални резултати особено очевадно е това при отчитане на критични ъгъл на атака. На този етап от настоящето изследване не може да се посочи най-подходящият турбулентен модел, но позовавайки се на проведеното числено изследване може да се заключи, че DES k-ω SST модел на турбулентност е най-ефективен при решаването на подобен род проблеми макар и с някои неточности предвид получения най-близък наклон на кривите за c_L и c_D в зависимост от приетите експериментални данни. Предполагаме, че неточностите, които са явни при отчитането на критичният ъгъл на атака могат да се дължат на разработеният двумерен числен 2D модел на крилен профил NACA 2412, който е по-подходящ за академични цели, не толкова за решаване на практически задачи. Разработеният числен подход може да се подобри по следните начини: 1) чрез увеличаване на размера на изчислителната област;2)с увеличаване на плътността на мрежата; 3) чрез 3 D моделиране на цялата излетна конфигурация.