ISSN: 0974-5823 Vol. 7 No. 1 January, 2022

International Journal of Mechanical Engineering

ON CFD WING PERFORMANCE FOR ULTRA-LIGHT AIRPLANE

Taha Ahmed Abdullah, Mohammed Saleh Mohammed
Mechanical Department, Engineering College, Mosul University, Mosul, Iraq

ABSTRACT

The CFD scheme will be used to study wing performance for ultra-light airplane at low speed. NACA 0012 airfoil is used to model the wing. The pressure coefficient distribution around the airfoil will be computationally obtained for the same speed and different angles of attack. The lift and drag coefficients are then calculated from the pressure coefficient. The results obtained in this paper prove good compatibility with other theoretical and experimental methods.
Keywords: Lift, Drag Pressure coefficients

1. Introduction

The wing is that important part of the plane that is responsible for generating lift to the plane when the plane reaches take-off velocity due to the flow of air on the upper and lower surface of the wing, where a large pressure difference is formed between the two surfaces of the wing resulting in the lift force required for the plane to take off.
A large number of researchers have worked in the field of studying different types of wings in terms of specifications, shape, and the urgent need for a specific type of this type, and we will mention a number of researchers in this paper.

The CFD tools, however, requires expensive computational resources for high Reynold numbers and large wing sizes. For this reason, the singularity method can play important role to predict the aerodynamics characteristics for high speed. Therefore, there is a need to measure the accuracy of the singularity method relative to the CFD and experimental data which provides an insight on the performance of the singularity method.

The aerodynamic characteristics of NACA0012 airfoil was extensively studied using CFD tools by [1] and [2]. Abdullah T. employed the singularity method by dividing the wing into a large number of panels and each panel was represented by linear strength vortex to model flow around the wing, the pressure coefficient was numerically calculated, and this leads to calculate the lift and drag coefficients and these results was compared with analytical and experimental data [3]. The singularity method was used to calculate pressure distribution coefficient around airfoil to correct experimental data from wind tunnel with porous walls [4]. Abdullah. et. al. [5] have used singularity method by dividing wing to sufficient number of panels to calculate pressure coefficient. The results were compared with experiments to correct the data obtained from wind tunnel with solid walls. Mragank et al [6] used FLUENT software package of ANSYS to evaluate lift and drag coefficients by simulation fluid flow around airfoil. Ankan Dash [7], numerically analyzed flow field around airfoil NACA 0012 at different angles of attack and compared the results with available experimental data. In reference [8] CFD simulation of airfoils is conducted using shear stress transport (SST) model to predict the lift and drag characteristics of the airfoil. Hess [9] is found the solution of flow field by simulating the surface by a number of panels, and solving algebraic set of linear equations to calculate the unknown strengths of the singularities.

2. Numerics

The physical domain is discretized using structured mesh employing sphere of influence which is centered on the middle of the airfoil, see Figures1, and 2. The two-dimensional, steady state, Navier Stokes equations are solved using pressure-based solver which is suited for low Mach numbers. The airfoil, upper and lower walls are assigned with no slip boundary conditions. Constant velocity is assigned for the inlet and zero relative pressure is given to the outlet boundary conditions. The grid size is refined until appropriate solution convergence is achieved.

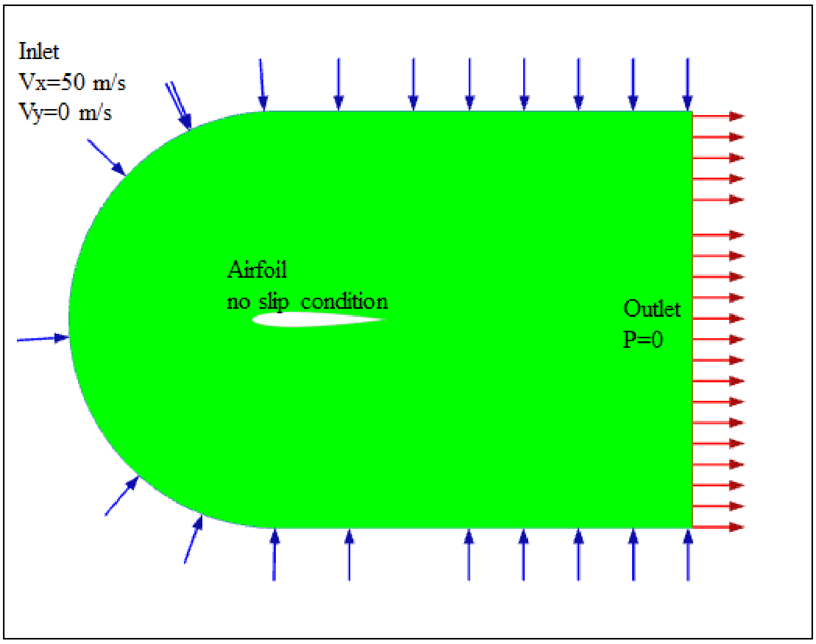

Fig.1. computational domain and associated boundary conditions.

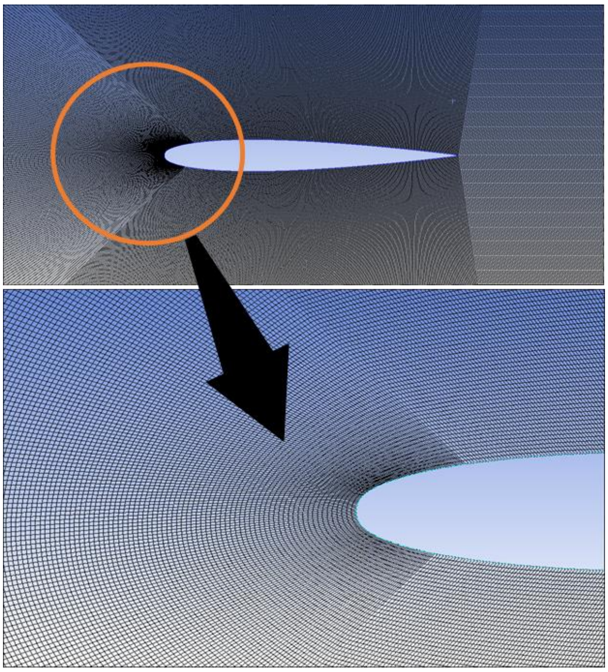

Fig.2. grid density

2.1. Boundary conditions

The research main point is to study the flow around an airfoil at various angles of attack (0°, 2°, 4° and 6°). The assigned boundary conditions are shown in the Table below.

Table 1 boundary conditions

No.	Parameter	Value
1	Flow stream velocity	50 m/s
2	Reference pressure	101325 Pa
3	Density	1.225 kg/m^3
4	Reynolds Number (Re)	10^6
5	Chord	1 m
6	Air stream temperature	288k
7	Angles of attack	0,2,4,6 degree

3. Results

3.1 Velocity Field

The magnitude of velocity contours is shown in Figures 3,4,5 and 6 for the different angles of attack 0o, 2o, 4o and 6o obtained from CFD simulations. From Fig.3, at zero angle of attack, it can be observed that the velocity is approximately equal at the upper and lower surfaces of the airfoil, so the lift force is equal to zero. When the angles of attack are increased as shown in Fig. 4,5 and 6, an increase in velocity occurs on the upper surface of the wing, while the opposite occurs on the lower surface of the airfoil with a decrease in velocity. The velocity at the stagnation point is almost zero at the leading edge of the airfoil, then the velocity begins to increase on the upper and lower airfoil surfaces, according to the change in the angles of attack.

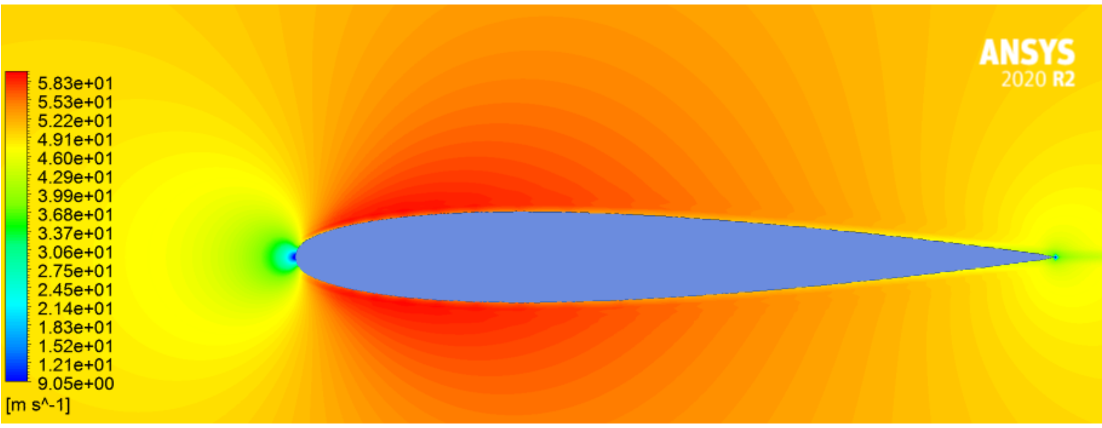

Fig.3. Velocity contour at angle of attack = 0

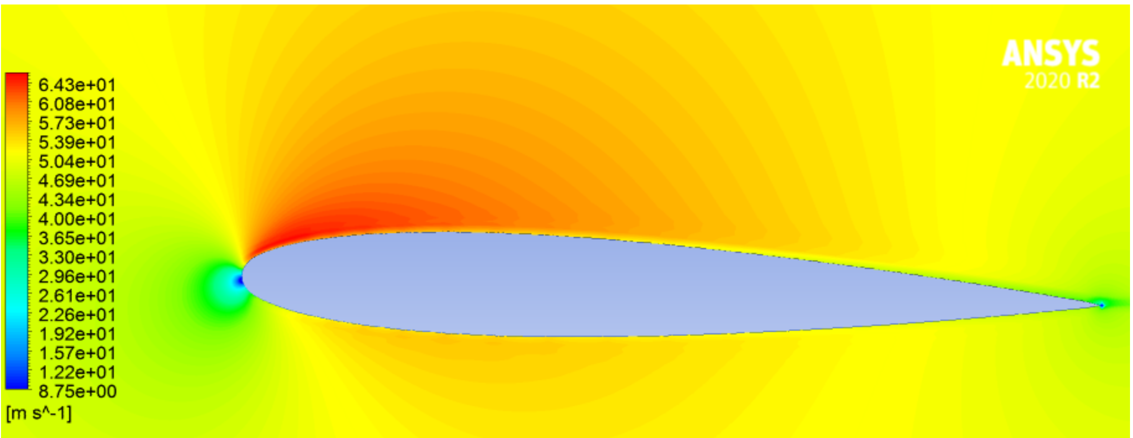

Fig.4. Velocity contour at angle of attack = 2°

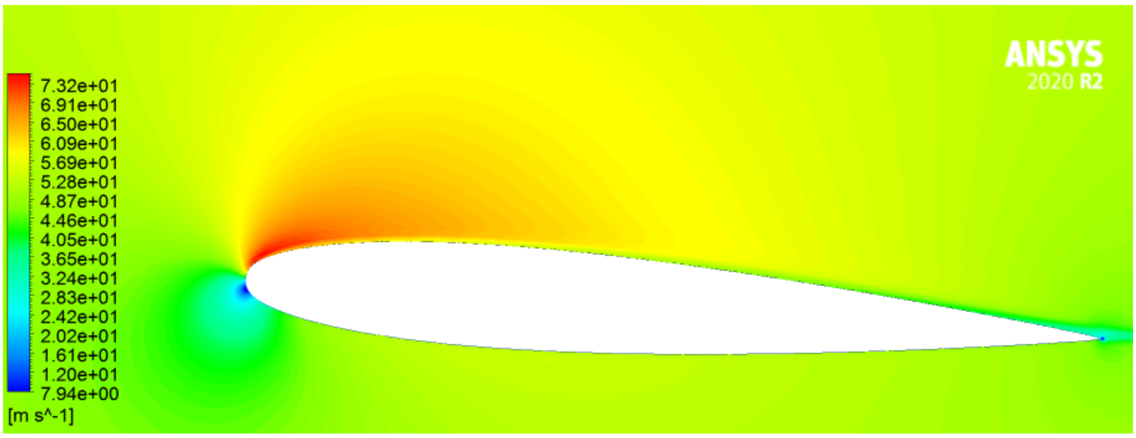

Fig.5. Velocity contour at angle of attack = 4°

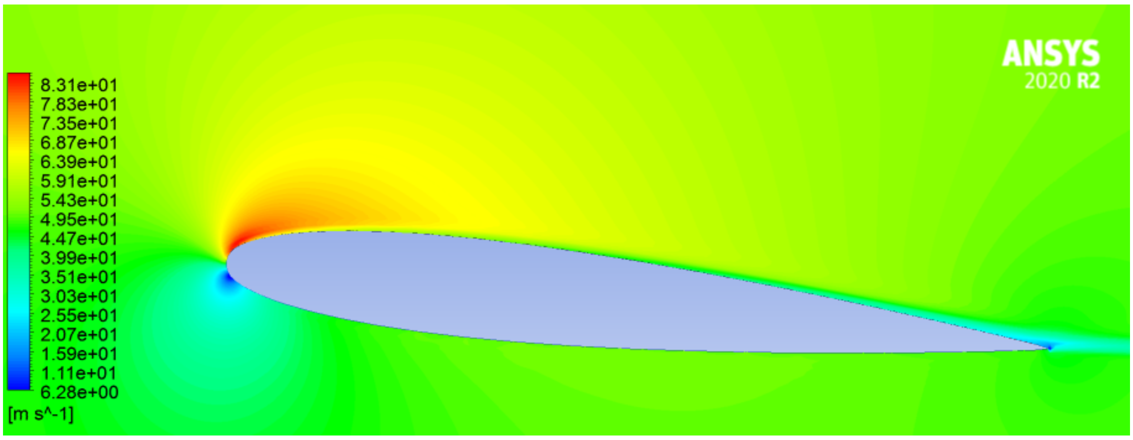

Fig.6. Velocity field at angle of attack = 6º

3.2 Pressure Field

The pressure value for the different angles of attack 0º, 2º, 4º and 6º obtained from CFD simulations as shown in Fig. 7,8,9 and 10. At the stagnation point on the leading edge of the airfoil, the value of the velocity is nearly zero, so the largest pressure formed at this point, which is called the stagnation pressure or the total pressure. At zero angle of attack, the distribution of pressure on the two surfaces of the airfoil is approximately equal, resulting in no significant lift force for the airfoil Fig.7. At angles of attack 2o, 4o and 6o, a pressure difference will be generated on the surfaces of the airfoil, causing a net force perpendicular to the airflow direction and this force is analysed into two components, one of them is a vertical force called the lift force and the other is a horizontal force called the drag force and the lifting force is greater than the drag force.

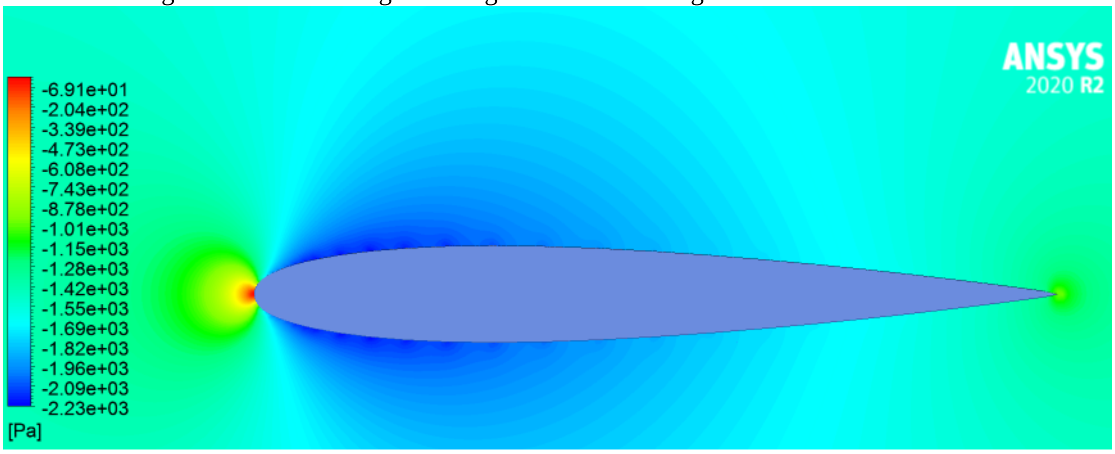

Fig.7. Pressure field at angle of attack = 0

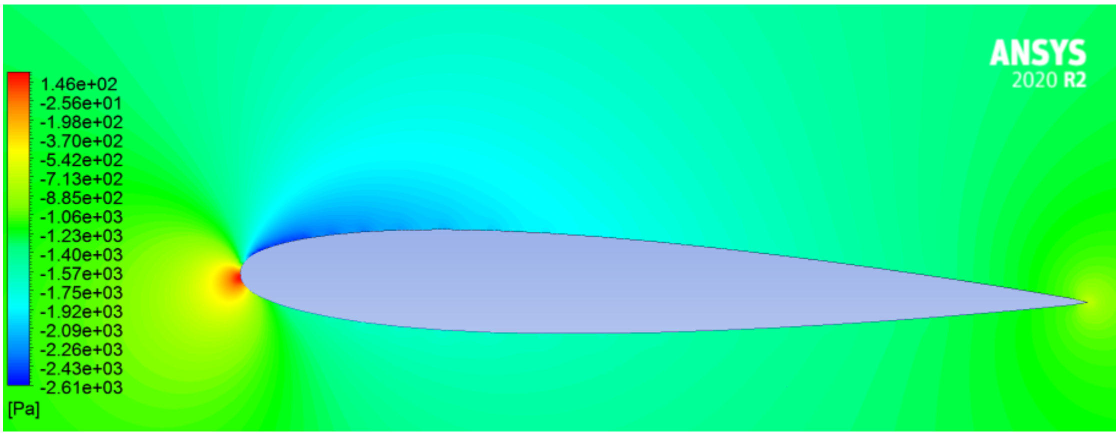

Fig.8. Pressure field at angle of attack = 2º

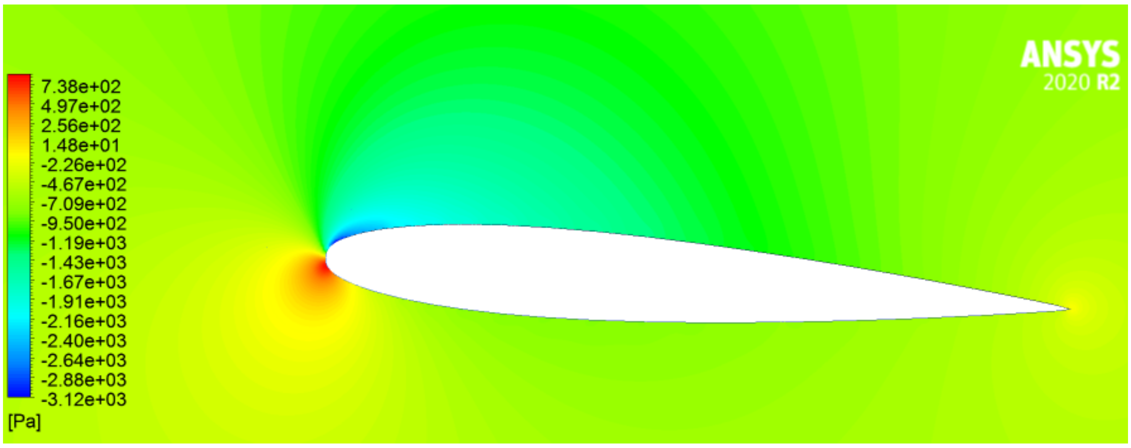

Fig.9. Pressure field at angle of attack = 4º

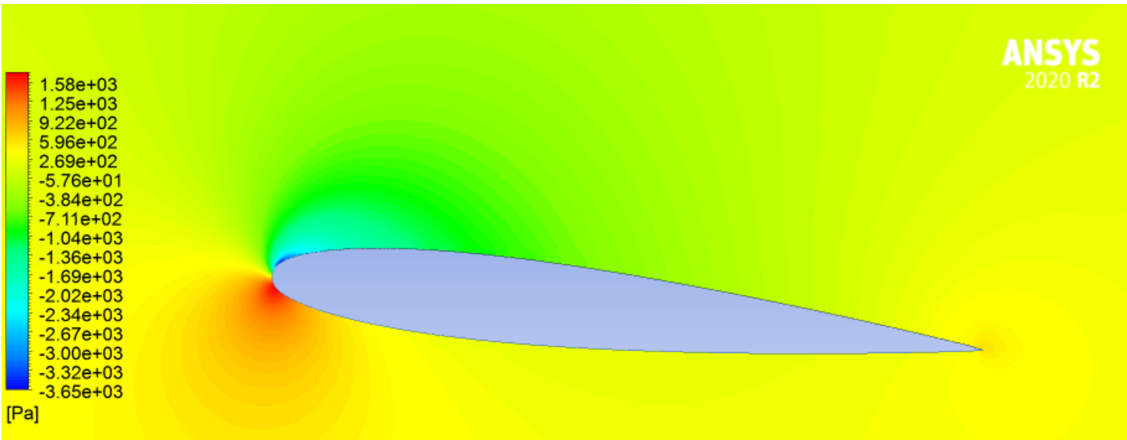

Fig.10. Pressure field at angle of attack = 6º

3.3 Distribution of pressure

The pressure field around NACA0012 airfoil for different angles of attack is depicted in the Fig.11. At zero angle, it is clear that the pressure distribution line of the pressure and suction sides are matched. and this proves that no lifting force is generated at this angle. With the increase in the angle of attack, the pressure distribution lines for the surfaces of the airfoil begin to diverge from each other to prove that the pressure difference between the upper and lower surfaces of the wing began to increase and as the angle of attack increased, the pressure difference increased and as a result the lift force required for the wing increased. As shown in Fig.11, it can be seen that the pressure distribution difference at the leading edge of the wing is large and this difference decreases significantly at the trailing edge, meaning that most of the lifting force is generated at the front part of the wing.

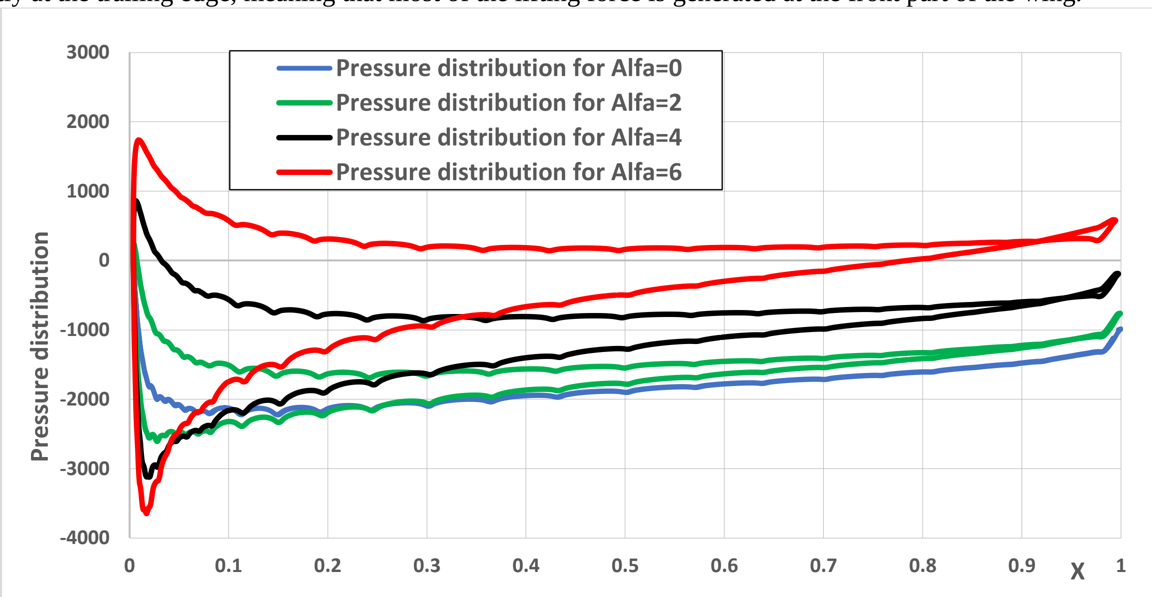

Fig.11. Pressure distribution

3.4 Lift and drag coefficients

The lift and drag coefficients for different angles of attack was calculated using inviscid model. The CFD simulations results were verified with experimental two wind tunnels data (T-38 and NASA) references [10], [11], numerical method [3] and analytical results from Abbott and von Doenhoff [12] and proved a good agreement between the different results

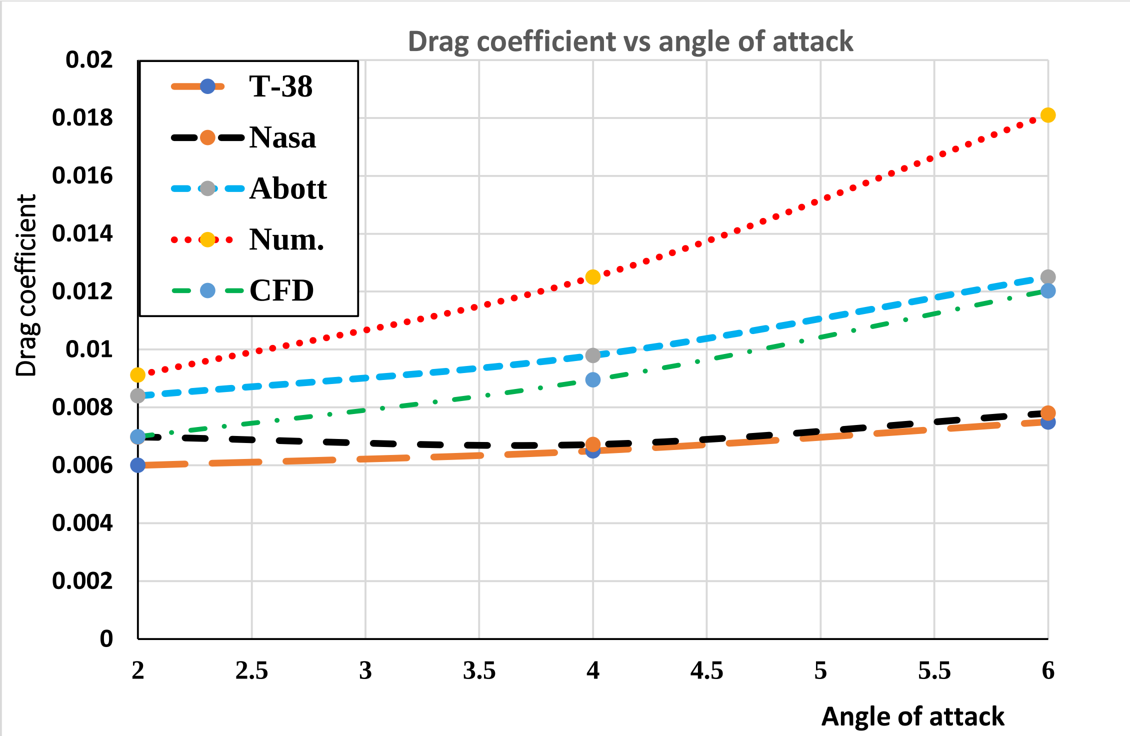

Fig.12. Drag vs angle of attack

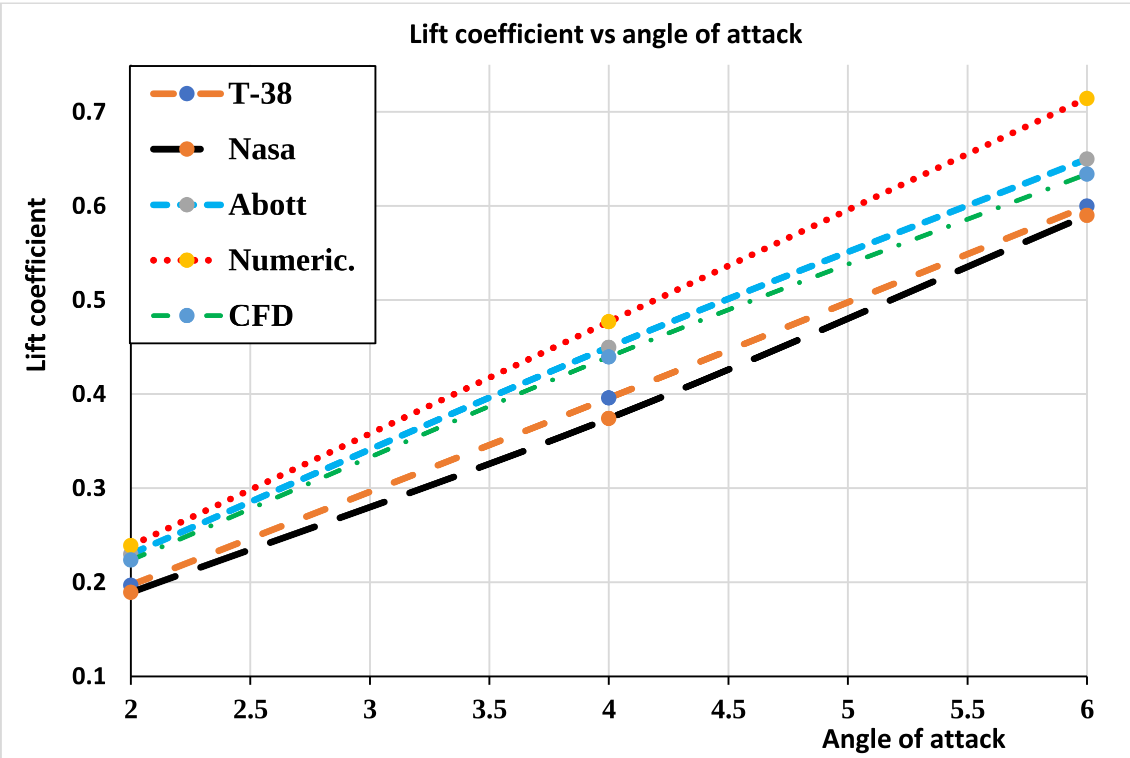

Fig.13 Lift vs angle of attack

4. Conclusions

A study of the aerodynamic performance of the airfoil 0012 for different angles of attack (0º, 2º, 4o and 6º) was carried out through the CFD software Ansys-Fluent using viscous inviscid model. There is a difference between the velocity and pressure distribution on the pressure and suction side of the airfoil, it is clear that the velocity on the upper surface of the airfoil is greater than that on the lower surface. This means that there is a difference in the pressure distribution on the surfaces of the airfoil so that the pressure distribution on the suction side of the airfoil is greater than on the pressure side. It leads to the generation of an upward force that is perpendicular to the direction of air flow and this force is analysed into two components, vertical is called the lift force, and the horizontal is called the drag force, and the lifting force is greater than the drag force. At zero angle of attack the difference in velocity and pressure distribution on the two airfoil surfaces will be so small that the lift force generated cannot be taken into account. The value of the lift and drag coefficients calculated by CFD analysis showed good agreement with the experimental and analytical methods.

References

[1] P.-C. S. Kapsalis, S. Voutsinas and N. S. Vlachos, "Comparing the effect of three transition models on the CFD predictions of a NACA0012 airfoil aerodynamics," Journal of Wind Engineering and Industrial Aerodynamics, vol. 157, pp. 158-170, 2016.

[2] L. Jiang and C. Liu, "Parallel DNS for Flow Separation and Transition around Airfoil," in Parallel Computational Fluid Dynamics 2003, Elsevier, 2004, pp. 205-212.

[3] Taha A. Abdullah, "Numerical Investigation on Wing Performance Using Singularity Method," International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), vol. 10, no. 3, pp. 7029-7040, 2020.

[4] Taha A. Abdullah and M. Mohammed, "Measurement Correction of Two-Dimensional Wind Tunnel with Porous Walls Using Singularity Method," " International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), vol. 10, no. 3, pp. 3715-2730, 2020.

[5] Taha A., Zlatko P., Zoran S., Ivan K. and Jovan. I., "Two-dimensional wind tunnel measurement corrections by the singularity method," Technical Gazette, vol. 22, no. 3, 2015.

[6] Mragank Pratap Singh, Dr. Mahendra Singh Khidiya and S. Soni, "CFD Simulation of an Airfoil at different Angle of Attack," IJSRSET, vol. 2, no. 3, 2016.

[7] A. Dash, "CFD Analysis of wind turbine airfoil at various angles of attack," IOSR Journal of Mechanical and Civil Engineering, vol. 13, no. 4, 2016.

[8] Tousif Ahmed, Tanjin Amin, R. Islam and &. S. Ahmed, "Computational study of flow around a NACA 0012 wing flapped at different flap angles with varying Mach numbers," Global Journal of Researches in Engineering, vol. 13, no. 4, 2013.

[9] J. L. Hess, "Panel Methods in Computational Fluid Dynamics," Annual Review of Fluid Mechanics, vol. 22, pp. 255-274, 1990.

[10] A. Vitić, "Experimental determination NACA 0012 characteristics in the T-38 wind tunnel two-dimensional test section," Scientific Technical Review, vol. XLI, no. 2, 1991.

[11] C. L. Ladson and A. S. Hill, "Pressure distributions from High Reynolds number transonic tests of an NACA 0012 airfoil in the Langley 0.3-meter transonic cryogenic tunnel," NASA Technical Memorandum 100526, 1987.

[12] I. H. Abbott and A. E. von Doenhoff, Theory of Wing Sections, New York: Dover Publications, 1959. [2] Luckring, J.M. and Hummel, D. (2008). *Chapter 24 – What Was* .