Aerodynamic Analysis of Aircraft Wings Using CFD

Kimay Gore[1], Abhijeet Gote[2], Aniket Govale[3], Ajit Kanawade[4], Shailesh Humane[5]

[1,2,3,4] B.E Mechanical, Shivajirao S. Jondhle College of Engineering, Maharashtra, India
[5]Asst. Professor, Dept. of Mechanical Engineering, Shivajirao S. Jondhle College of Engineering, Maharashtra, India

---

Abstract - *Airfoil is a streamline body which plays an important role in any aircraft because it has to generate adequate amount of lift to hold the aircraft in the air with less amount of drag. The design of an airfoil shape, with desired aerodynamic characteristics is difficult till date. In early days the design was randomly set and it was tested in a flow section, then Wright Brothers came up with cambered section. NACA has given a proper definition for airfoil which helps us to generate airfoil using formulas and not randomly. In this work a detailed study of NACA airfoil is done at Various Angle of Attack (ie.0, 8, 16, 20, 25, 27) and at the same Mach No. 0.6. The lift and drag coefficient for airfoils are obtained by analyzing the pressure distribution on the airfoil surface. The simulations were computed in the software ANSYS 16.2. Lift increases as the approach ascends from 0 to 25 degrees and at a specific point, most extreme lift is produced. On the further increment in the angle of attack, drag turns into the overwhelming component and the area enters the stall mode.*

This approach is to check the NACA 4412 Aerofoil which is used in Aircraft Wing. The results showed that the NACA 4412 Aerofoil is getting maximum lift at 25^o and there after the lift force decreases.

***Key Words*: Airfoil 4412, Ansys 16.2, Computational Fluid Dynamics, Angle of Attack, Lift, Drag, Mach No.**

1. INTRODUCTION

An Airfoil is a cross sectional area which is used in aircraft wing design. Its fundamental concept is to offer lift to a plane by reducing the drag to a sufficient amount. The measurement of forces and coefficient of lift depends upon configuration and assembly of various parts of the desired aircraft. Heavier one has more lift while lighter one has less lift contrasted with heavier ones. In like manner, dependent upon the use of plane, Airfoil cross section is settled. Lift however apply extra forecast to the raising velocity of the aircraft, which in turns relies upon the plane with respect to the flight speed. Henceforth, the coefficient of lift and weight is the central factor to make indicate how the lift reacts according to the speed and different parameters.

Airplane wings which are level and vertical stabilizers, helicopter rotor cutting edges, propellers, fans, compressors turbines all have Airfoil plans. Indeed, even sails, swimming and flying creatures utilize Airfoil. An airfoil molded ribs is adequate to chop down the force on an automobile parts or other motor gadgets in order to enhance the adhesive friction known as traction.

A wing following the laminar stream has a biggest partiality of thickness in the middle piece of camber line. It shows a negative weight slant along the stream and has a similar effect when we dissect the Navier– Stokes numerical explanations in the straight organization. So if we keep up most prominent camber in the center, a laminar stream over a greater rate of the wing at a higher speed can be accomplished. In any case, with particles on the wing, this does not work. Since such a wing backs off more successfully than other.

1.1 Aircraft Wing Design

The determination of right wing is the most vital part of plane outline which decides lift compel age, drag constrain age, slow down point. Delta wings discovers its application for flying at supersonic speed and henceforth utilized for Fighter flies and space transports. Delta wings additionally give advantages of cleared wings (diminished drag at supersonic paces) because of their high scope, and they are basically effective and give an extensive inside volume which can be utilized for fuel tanks. They are likewise moderately basic and cheap to make. Now in the outline procedure CFD investigation plays a pivotal, if not its most essential, part. Wind burrow models are by and large extremely costly to manufacture, costing maybe a huge number of dollars or more, and wind burrow test time is a noteworthy cost driver amid a task. In this work we chipped away at CFD Analysis of delta wing of a Fighter flying machine that is F-16 (Felcon Fighter) which is fundamentally utilized by USA. We dissect the wing of this flying machine on CFD.

Computational liquid elements, normally shortened as CFD is a branch of liquid mechanics that utilizations numerical techniques and calculations to take care of and investigate issues that include liquid streams. PCs are utilized to play out the computations required to mimic the connection of fluids and gases with surfaces characterized by limit conditions.

The central premise of all CFD issues are the Navier–Stokes conditions, which characterize any single-stage (gas or fluid, however not both) liquid stream. These conditions can be improved by expelling terms portraying gooey activities to yield the Euler conditions. Encourage rearrangements, by evacuating terms portraying vortices yields the maximum capacity conditions. At long last, for little irritations in subsonic and supersonic streams (not transonic or hypersonic) these conditions can be changed to yield the changed potential conditions.

1.2 Aerodynamics

Aerodynamics is basically the branch of science which is nothing but the study of motion of air and the way an object moves through the air.

The concept of Aerodynamics was brought into effect by the Wright Brothers in early 1900s by bringing man's desire to fly in reality. The concept of Aerodynamics has contributed hugely in all the types of Automobiles and the making of Fighter planes, high speed trains, supercars and has been scrutinized to a different level.

Use of Aerodynamics with the help of mathematical study, formulae, approximations, wind tunnel experiments and computer simulations, has formed a scientific basis for the ongoing developments.

The two major branches of Aerodynamics are –

- **Incompressible Aerodynamics –** It is concerned with the incompressible flow, which is the stream in which density remains constant in time and space. The Subsonic (or the low-speed) flow dynamics is the further division of the incompressible Aerodynamics.
- **Compressible Aerodynamics –** It is concerned with the compressible flow, which is the flow in which the change of density with respect to pressure is non-zero along a streamline.

The further divisions of the compressible flow are –

(1) Sub-Sonic Flow – When the Mach number is less than Unity.

(2) Sonic Flow – When the Mach number is equal to Unity.

(3) Supersonic Flow – When the Mach number is between 1-6.

(4) Hypersonic Flow – When the Mach number is more than 6.

2. NUMERICAL SIMULATION

2.1 Computational Fluid Dynamics and Analysis

CFD is a numerical strategy used to reproduce physical issues with use of liquid examinations. This strategy is used to inquire about arrangement without making a physical model and can be a vital to grasp properties of new mechanical plans. By using a recreation instead of doing lab tries, one may get comes about quicker. An indispensable thing in the use of CFD is to grasp the changes in programming, and know the requirements. Notwithstanding the way that the CFD programming utilizes without a doubt comprehended speaking to correlations, genuine separations are made similar to grid and speaking to geometries.

3. METHODOLOGY

General

The process of the numerical simulation of fluid flow using the above equation generally involves four different steps and the details are given below.

(a) **Problem identification.**

1) Setting the modelling ends.
2) Placing the model to domain.

(b) **Pre-working.**

1) Creating an airfoil model.
2) Meshing configuration.

(c) **Solver.**

1) Demonstrating the physics
   - Representing the flow (e.g. turbulent, laminar etc.)
   - Placing the appropriate boundary condition.
2) Using different numerical strategy to discretize the governing equations.
3) Checking the convergence by iterating the equation till precision achieved.
4) Figure out the Solution by Solver Setting.
   - Initialization.
   - Solution Control.
   - Monitoring Solution.

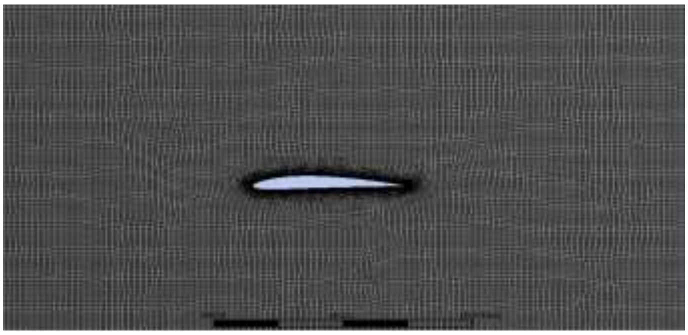
(d) **Post processing**

1) Analyzing the results.
2) Graphical diagrams.
3) Contour Details.

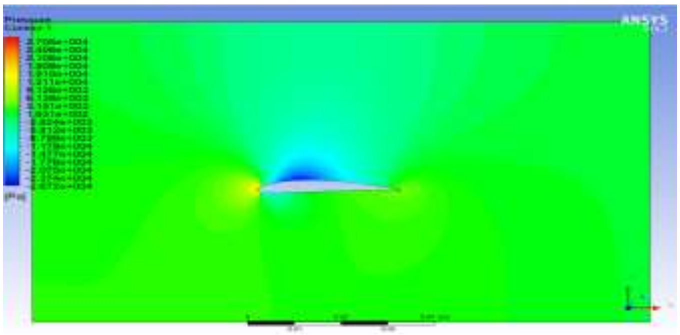
4. RESULTS AND DISCUSSION

1) NACA 4412 at 0 degree angle of attack:

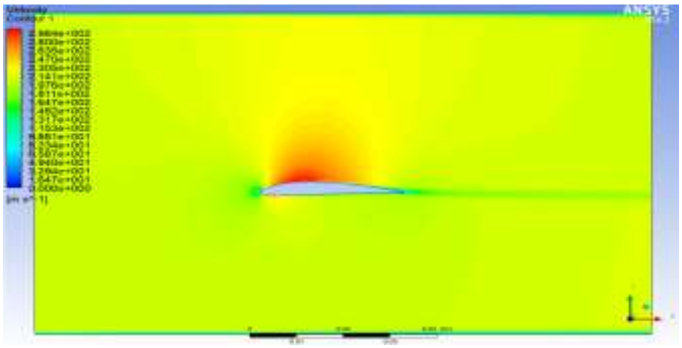
a) Mesh



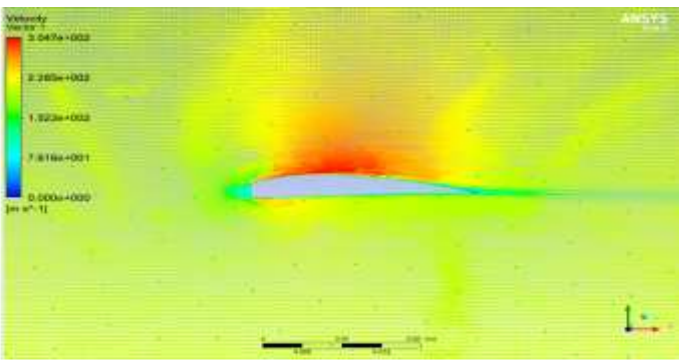
b) Pressure Contour Diagram:



c) Velocity Contour Diagram:

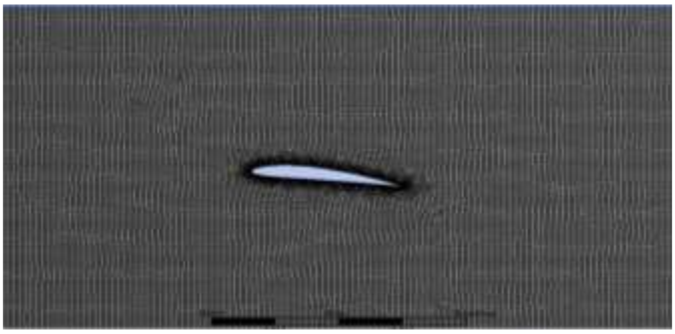


d) Velocity Vector Diagram:

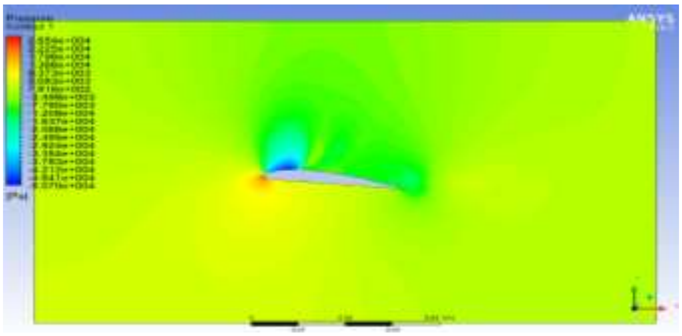


2) NACA 4412 at 8 degree angle of attack:

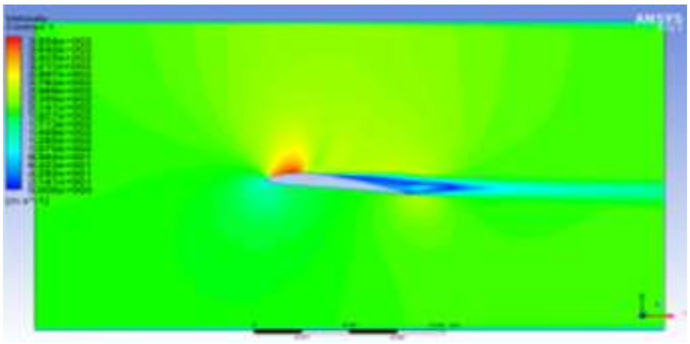
a) Mesh



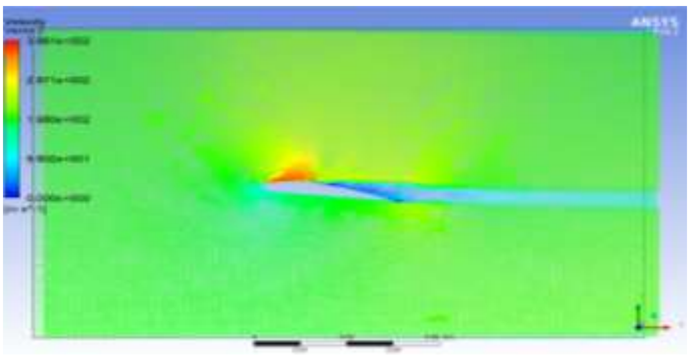
b) Pressure Contour Diagram:



c) Velocity Contour Diagram:

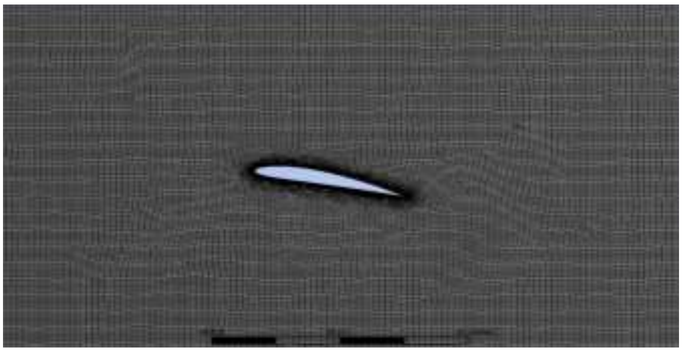


d) Velocity Vector Diagram:

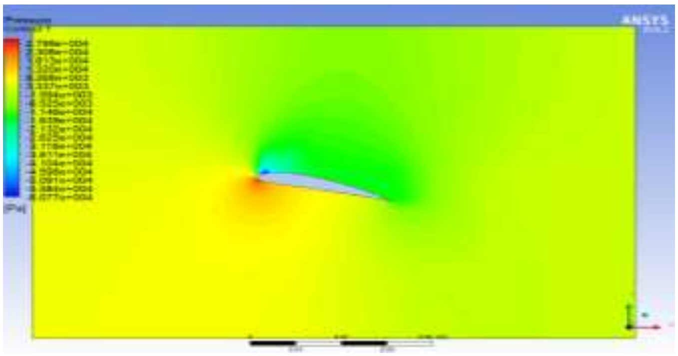


2) NACA 4412 at 16 degree angle of attack:

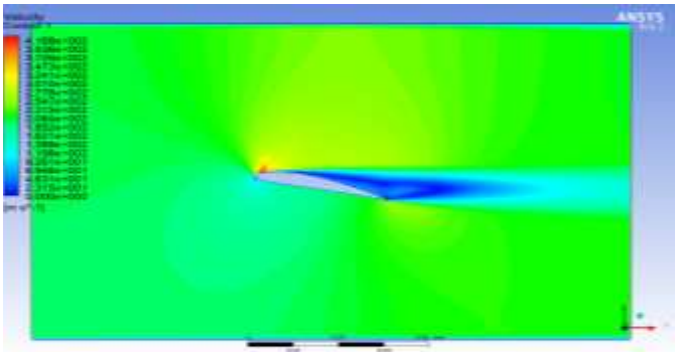
a) Mesh



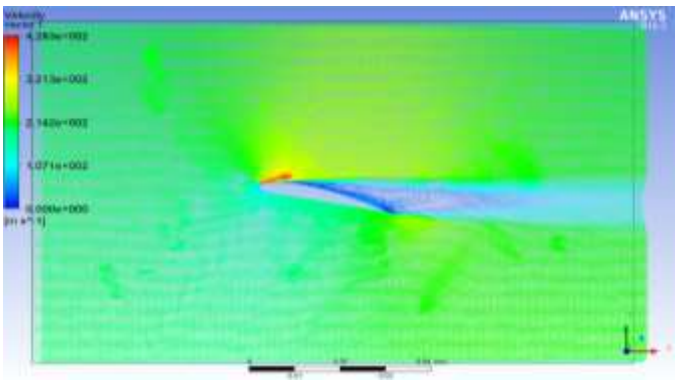
b) Pressure Contour Diagram:



c) Velocity Contour Diagram:

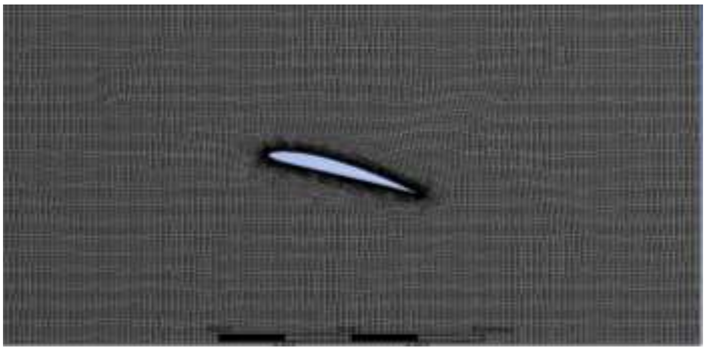


d) Velocity Vector Diagram:

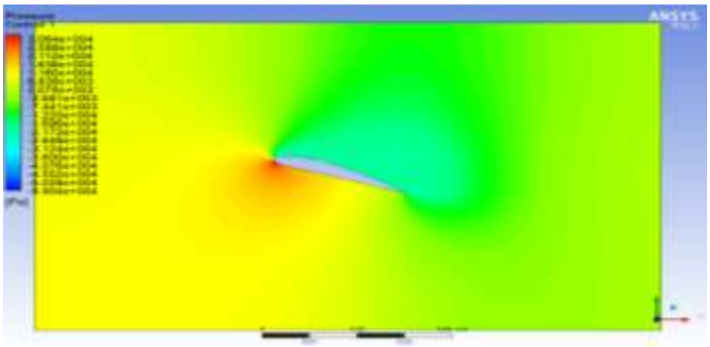


3) NACA 4412 at 20 degree angle of attack:

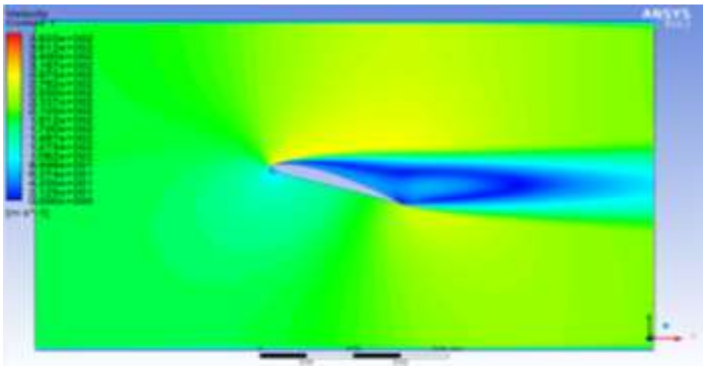
a) Mesh



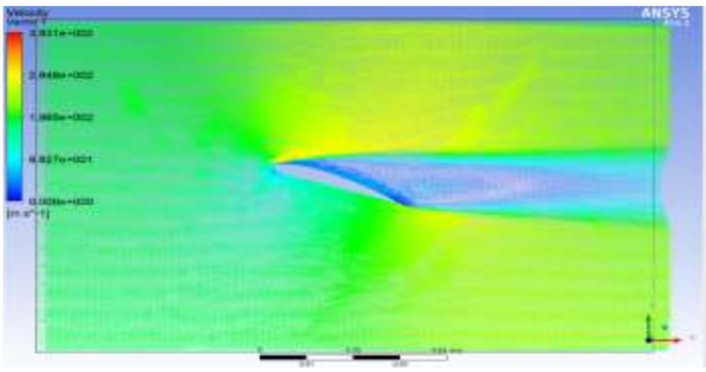
b) Pressure Contour Diagram:



c) Velocity Contour Diagram:

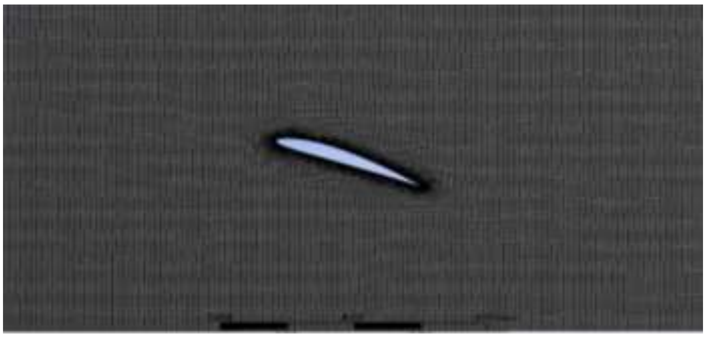


d) Velocity Vector Diagram:

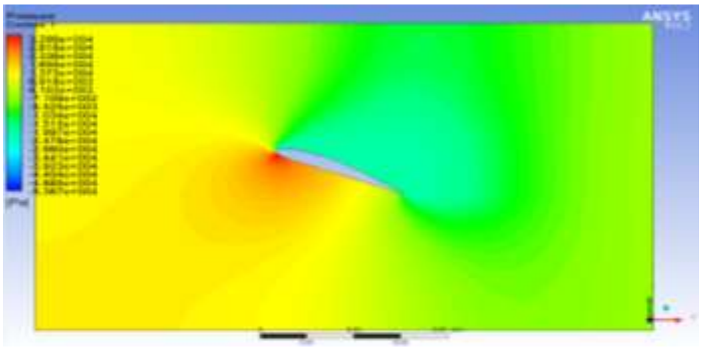


3) NACA 4412 at 25 degree angle of attack:

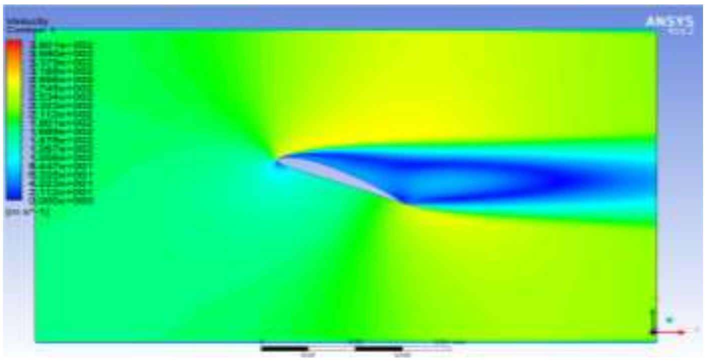
a) Mesh



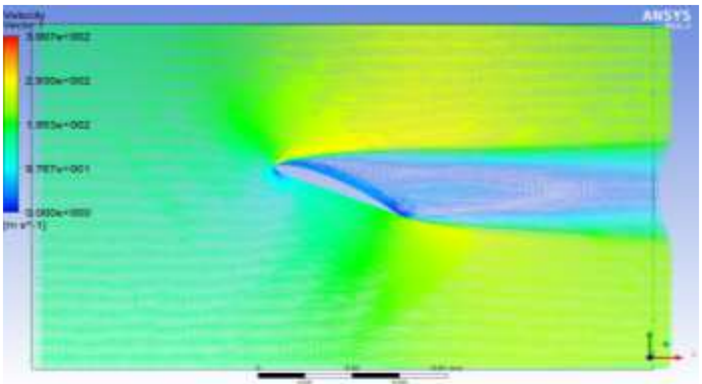
b) Pressure Contour Diagram:



c) Velocity Contour Diagram:

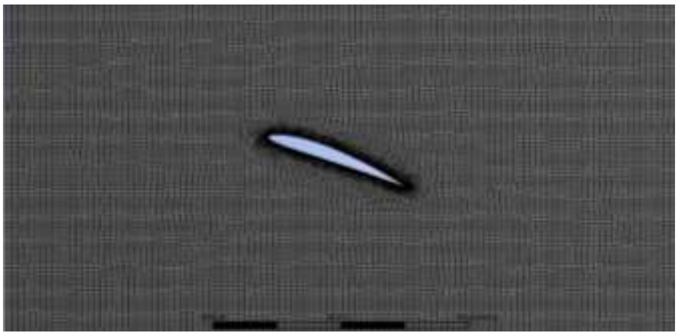


d) Velocity Vector Diagram:

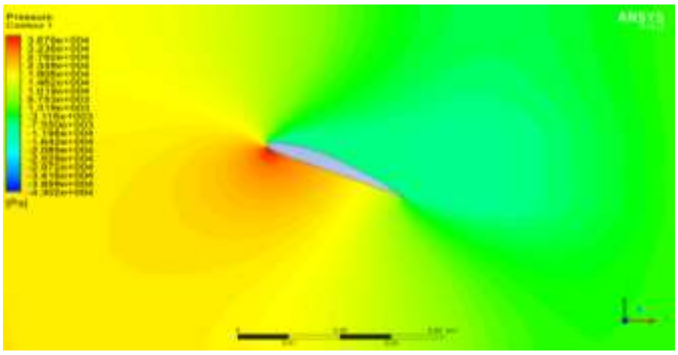


4) NACA 4412 at 27 degree angle of attack:

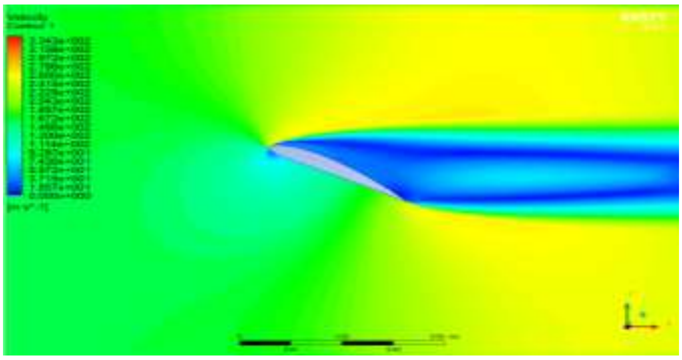
a) Mesh



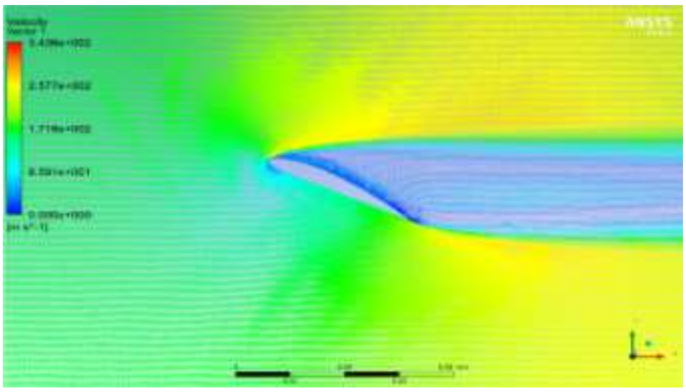
b) Pressure Contour Diagram:



c) Velocity Contour Diagram:



d) Velocity Vector Diagram:



4.1 Comparison of Final Results

Table -1: Comparison of final Readings.

Angle of Attack (Degree)	Velocity (m/s)	Lift coefficient	Drag coefficient	Lift force (N)	Drag force (N)
0	208.25	1.5863e-02	5.2905e-04	404.75	13.49
8	208.25	2.6890e-02	1.7887e-03	686.10	45.64
16	208.25	3.3212e-02	5.6749e-03	847.43	144.79
20	208.25	3.6594e-02	1.2795e-02	933.72	326.48
25	208.25	3.9175e-02	1.7996e-02	999.56	459.17
27	208.25	3.2040e-02	1.6002e-02	817.51	408.30

5. CONCLUSION

1. Coefficient of lift will be higher at an angle of 25 degree.

2. As the angle of attack increase the amount of lift created by the airfoil also increases.

3. A lowering of pressure on the upper surface results in developing pressure gradient. 4. NACA 4412 Airfoil can be used in Aircraft Wing. 5. The lift Force is considerably large then the Drag Force.

6. FUTURE SCOPE

- Improvement of the modern aircraft.
- Suggestion of New Configuration.
- Suggestion of New rules and requirement.

REFERENCES

1) Nguyen Minh Triet, Nguyen Ngoc Viet, Pham Manh Thang "Aerodynamic Analysis of Aircraft Wing "VNU Journal of Science: Mathematics –Physics, Vol. 31, No. 2 (2015) 68-75.

2) Ankan Dash "CFD Analysis of Wind Turbine Airfoil at Various Angles of Attack", IOSR Journal of Mechanical and Civil Engineering, Vol No.13, Issue 4(2016)18-24.

3) Ankit Ahuja, "CFD Analysis of an Aerofoil, International journal of engineering research, Vol. No.3,Issue No.3(2014)154-158.

4) Mayurkumar kevadiya"2D Analysis of NACA 4412 AIRFOIL", International Journal of Innovative Research in Science, Engineering and Technology Vol. 2, Issue 5, May (2013)1686-1691.

5) S.Kandwal" Computational Fluid Dynamics study of Fluid Flow and Aerodynamic Forces on an Airfoil", International journal of engineering research & Technology Vol No.1, Issue 7, (2012)1-8.

6) Web Reference http://www.airfoiltools.com

7) A Textbook on "An Introduction to Computational Fluid Dynamics- The Finite Volume Method "–H. K. Versteeg and W. Malayasekera. Published 1995 by Longman Scientific and Technical.

BIOGRAPHIES

Mr. Kimay B Gore
He was born in Kalyan Maharashtra, India in 1996.He completed his HSC with First Class Division in 2012-2014 and pursuing B.E Degree in Mechanical Engineering from Shivajirao S Jondhle College, Maharashtra, India, in 2014-2018.

Mr. Abhijeet R Gote
He was born in Dombivli, Maharashtra, India in 1996.He completed his HSC with Distinction in 2012-2014 and pursuing B.E Degree in Mechanical Engineering from Shivajirao S Jondhle College, Maharashtra, India, in 2014-2018.

Mr. Aniket B Govale
He was born in Kalyan, Maharashtra, India in 1997.He completed his HSC with in First Class Division 2012-2014 and pursuing B.E Degree in Mechanical Engineering from Shivajirao S Jondhle College, Maharashtra, India, in 2014-2018.

Mr. Ajit S Kanawade
He was born in Thane, Maharashtra, India in 1996.He completed his HSC with in First Class Division 2012-2014 and pursuing B.E Degree in Mechanical Engineering from Shivajirao S Jondhle College, Maharashtra, India, in 2014-2018.