International Journal of Research in Engineering, Science and Management
Volume-3, Issue-2, February-2020
www.ijresm.com | ISSN (Online): 2581-5792

CFD Analysis of Airofoil section of NACA 0024 Lift and Drag at Different Angle of Attack

Alok Ranjan
M.Tech., Dept. of Mechanical Engineering, Goel Institute of Technology and Management, Lucknow, India

***Abstract*: Aerofoil is used throughout the world in the wider authenticity of the industries. Some important applications are in the Steam turbine, ventilation system, electric motor and generator cooling, air craft, turbines and many industrial processes.**

Many researchers and engineers have been trying their best to design the aerofoil in the most effective manner to meet the special requirement of application. The criteria for the aerofoil's cost, ease of construction, efficiency and conservation of energy, other ideas are also being considered in the design. We know very well, all aerofoil are axial flow fan. In many studies, there are various researchers available in analyzing and simulating axial and centrifugal fans. The axial flow fans have also been designed and signed by researchers. The design of various blade sections of the display of axial flow fans and axial flow fans has been simulated in an experimental or numerical form. The present work includes numerical study of axial flow fan section of aerofoil. The purpose of the study is to simulate the flow characteristics around the specially designed aerofoil for the nine different values of the striking angle or angle of attack between 2 to 18 at 2 alternate. In this study "NACA0024" used. The result is the use of FLUENT as the pressure distributions and turbulence at the leading edge moving across, trailing edge and across the sections of aerofoil. On the basis of pressure distribution and turbulence, we justified lift and drag ratio because if lift and drag ratio is increase than efficiency of aerofoil is increases. Pressure distribution and turbulence has also been shown for nine cases and comparison of lift and drag show in chart.

***Keywords*: NACA0024, Axial flow fan, Aerofoil, Angle of attack, FLUENT, Lift and drag, CFD.**

1. Introduction

Energy is very important for human life as it is famous, energy is produced from fossil fuels, but there are two problems in fossil fuels. First of all, its resources are very limited. Second, they carry environmental pollution. For this reason, renewable energy emerges as an alternative resource. One of the wind energy from renewable energy is the wind turbine used to convert wind energy into electrical energy, but wind turbine efficiency is not good. Because of this, many scientists are tested on wind turbines and wind turbine parameters. One of the most important parameters of the wind turbine is the wing because the winds and wind energy are changed from wings to mechanical energy due to the wings. In literature, the profile of wings is called air film. The airfoil profile is an important parameter for wing design, because the wing efficiency increases depending on the air profile, so there are many studies on the aerofoil profile in the literature as numerical and experimental.

Experimental investigation is very important because of accuracy, however, those people find very time and money and whenever we want to change a parameter about our study, it is very difficult due to the time and economic fortunately, the investigator computational Very fast and easy to study fluid mobility (CFD) programs. These programs can give the right results in the form of experimental methods. In addition, according to the experimental methods CFD programs can be contributed in time and speed. The basic purpose of a "fan" is to transfer gas or steam on a large scale to the desired velocity. The pressure of the gas in the fan rotor or impeller is slightly higher for achieving this purpose. However, its main purpose is to move air or gas without any significant increase in its pressure. The total pressure developed by the fan is a sequence of gauges of some millimeter water.

Fans are used throughout the world in the wider authenticity of the industries. Some important applications are in the Steam Power Station, ventilation system, electric motor and generator cooling, and many industrial processes.

Fans play an important role in making cold effects through the heat exchanger. The efficiency of the machine depends mostly on the cooling effect. The cooler the machine can be, the more efficient the machine, and therefore the proper design selection of the fan in the heat exchanger is very important. Heat exchangers are condensers and evaporators, which are used mostly in air power units, refrigerators, boilers and condensers in most thermal power plants. The heat exchanger used in the automobile is radiator and oil cooler. The heat exchanger is used mostly in chemical and other industries. In its simplest form, axial flow fans are made of rotor made of blades planted for hubs in the phase. When it is rotated by an electric motor or any other drive, a flow fans.

A. Fans

Fan ventilation and air for industrial process requirements. Fans create a pressure in the fan system to transfer the air (or gas) against the resistance due to the defects, dampers, or other components. The fan rotor receives energy from the rotating shaft and transmits it in the air. There are two general classifications for fans: centrifugal or radial flow fans (see ED-

2400) and propeller or axial flow fans pass through the air impeller in a broader sense. The propeller or axial flow fan moves the wind in an axial direction with the tangent motion rotating made by the curved impeller blade. In a centrifugal fan, partially enters the air impeller and is accelerated by the blade and distributes the triangular. Axial flow fan increases the velocity of air through the dynamic or tangent force, which produces the velocity pressure (VP), the kinetic energy, the potential increase in the constant pressure (SP), the probability of the air generated by the potential energy centrifugal fan centrifugal force Generates energy (IP) in the rotating column of production, as well as the speed provided by the rotation (tangent) wind Energy (VP) leaves the blade tip production.

B. Axial flow fan

The word "axial flow fan", like "Radial Flow Fan", is derived from the main flow path through the rotor. According to the rotor rotation is in the path of the axis, the rotor of a hub, which is applied to the air foil in such a way that all the particles are given an increase in energy and the inevitable damage is kept as low as possible. In the general application, the fan, according to fig, becomes "the armor of the vessel". Axial flow from its inception in a corps is a simple design. The reason for this is that due to the axial flow path, originally part of the outer channel.

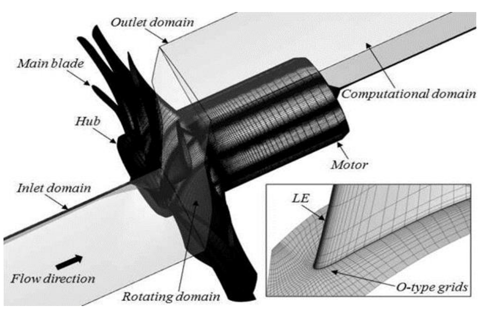


Fig. 1. Axial flow fan

In the general application, the fan, according to fig, becomes "the armor of the vessel". Axial flow from its inception in a corps is a simple design. The reason for this is that due to the axial flow path, originally part of the outer channel.

C. Type of axial flow fan

- Propeller Fans:
- Tube axial fans
- Vane axial fan
- Two-phase axial fans
- Multi-phase axial flow fan

D. Aerofoil

Feathers of a fixed wing plane, horizontal and vertical stabilizers, are made with an airflow-shaped cross section, such as helicopter rotor blades. Airfoil is also found in propellers, fans, compressors and turbines. PAL is also an Air Force, and under the surface of the sailboats, such as the scent board and keel, are similar in cross- sections and operate on the same principles as Airflows. Swimming and flying creatures and even many plants and algebra organisms employ air / matter / hydro aphelia: common example bird feathers, fish bodies, and sand-shaped size. An airfoil- shaped wing can make down fine on an automobile or other motor vehicle, can improve traction. Any object, moving the fluid, such as a flat plate, a building or brake deck attack, makes the aerodynamic force (lift) straight. Air force are more efficient lifting shapes, capable of generating more lift (up to a point), and able to generate lift with fewer drag.

The lift and drag curve obtained in the wind tunnel test is shown on the right. The curve represents an airflow with positive chamber, so some lift is produced at the zero angle of the attack. With the increase of the angle of attack, the slope of the lift curve increases broadly in the linear relationship, approximately 18 degrees on this airflow stalls, and the lift falls quickly, the fall in lift explained by the action of the upper surface border layer Which can be separated on the upper surface and becomes more in the back part of the stove angle. The displacement thickness of the layer of thickness changes to the effective shape of the airfoil, in particular it reduces its effective cement, by which the overall flow area is modified so that circulation and lift can be reduced. Thicker border layer also causes large increase in pressure, so that the overall drag becomes closer to the speed and crosses the stall point.

Aerofoil designs are the major aspects of aerodynamics, various air form serves different flight systems. Asymmetric aerofoil can generate elevators at the zero angle of the attack, while a symmetrical AF can be in the form of an aerobatic airplane, often in reverse flight. A symmetric airfoil can be used to increase the range of angle of attack to avoid spin-stalls in the area of the aileron and one wing tip. Thus, a large range of angles can be used without the different parts of the borders. A round of the subsonic airway is the leading edge, which is naturally insensitive from the angle of attack. The cross section is not strictly circular, however: before reducing the possibility of boundary layer separation, the radius of curvature increases after receiving the wing maximum thickness. It enhances the wings and takes the maximum thickness back from the edge of the edge Supersonic airfoil is much more angular in shape and can be a sharp edge edge, which is very sensitive to the angle of attack. The maximum thickness of a supercritical airfoil is near its leading edge, which takes a lot of time so that the supersonic flow can again get shock at subsonic speed. Typically, such a transnational airway and supersonic airfoil have a low number of camber, which reduces drag disruptions. Modern aircraft can have different airflow sections with wing periods in wings; each section of each wing is suitable for situations. Movable high-lift devices, flaps and sometimes slots, are fit for airfield on almost every aircraft. A backward flap works like an angler; However, it can be withdrawn partially in the wing if not used, unlike an aileron.

There is maximum thickness in the middle chamber line between a laminar flow wing. Analyzing the Navyer-Stokes equation in linear governance shows that negative pressure with flow is similar to reducing the slope speed, with the maximum camber in between, laminar flow over a large percentage of the wings at high speeds It is possible to maintain However, some surface contamination will obstruct the flow of laminas, which will cause it to become turbulent. For example, with the rain on the wings, the flow will become turbulent, under certain conditions, the insect debris on the wings will affect the loss of small areas of laminar flow too. [5] Prior to NASA's research in the 1970s and 1980s, the aircraft design community was understood by the efforts of application in the WW II era that laminar flow wing designs are practical using practical stamina and surface imperfections. After the new belief methods were developed with the overall contents (for example, graphite fiber), this belief changed and machining metal methods were introduced.

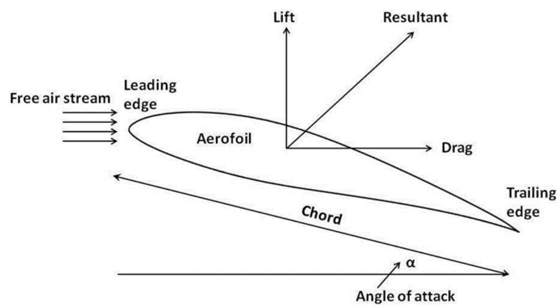


Fig. 2. Aerofoil section

2. History of aerofoil development

The first serious work started on the development of the airfoil class at the end of 1800s. Although it was known that when set at an angle of events, flat plates produce lift, some people suspect that the figures with curvature, which more closely match the bird wings, produce more lift or perform more efficiently. H.F. Philips patented a series of air film size after testing in one of the first air tunnels in 1884, in which "artificial currents of air were induced by a steam jet in a wooden trunk or gutter." Octave Chanute writes in 1893, "It seems very desirable that scientific experiments will be carried out on the skeletal surface of scientific shapes, because it is not impossible that the difference between the failure of the proposed flight machine and the failure on this Depending on the surface of an aircraft and the effect of a well-curved effect to obtain a 'maximum lift'. "At about the same time Otto Lilly Were. After carefully measuring the size of the bird wings, he tested his 7-meter diameter "Whirling Machine" with his Avifil Down (1894 book, "Bird Flight Age Beige of Aviation"). Lelynthal believed that the key to successful flight was the wing curvature or the camber. They also experimented with different nasal radius and thickness distribution. Airfoils used by the Wright Brothers are closely related to the category of Lilyanth: Thin and excessive cameras were quite likely because the initial tests of airflow class were done at very low Reynolds, where such classes behave better than the dense ones. Are there. The wrong impression is that efficient airflows were thin and very long; this was one of the reasons that the first airplane was biped. The use of such classes gradually decreased in the next decade. A wide range of Air Force was developed, which was primarily based on trial and error. Some of the many successful sections like Clark Y and Gottingen were used in the beginning of 1920 as the basis for families of classes tested by NACA.

A. Design and general construction

- *Motor enclosure* - Fan motor enclosure must be fully enclosed type.
- *Rotor* - The rotor of the fan motor will be well balanced.
- *Blade* - The fan will be fitted with two or more well balanced blades made of metal or other suitable material. Blades and blades will be fixed safely so that they are not loose in operation.
- *Growing* - Fans are members to be mounted against an outer wall in the given way to secure the mounting fan cover, they may be able to prevent the entry of rain water at the point of attachment.
- *Guards* - Properly designed guards will be provided by supply and supply on request and upon request, and to prevent accidental contact with rotating blades, inlet or outlet side or both will be fit. Guards will be attached safely and sufficiently hard to resist accidental contact with the blade.
- *Impact* - If necessary, submit information about the type of effect and instructions for investigation, investigation, their appropriate lubrication.

3. Methodology

A. Aerofoil terminology

Many words are used to describe aerofoil (dynamic flight, 2002).

Leading Edge = Ahead of Aerofoil

Trailing edge = Effort edge of aerofoil

Wire = line leading and linking the trailing edge. Indicates the length of aerofoil's

Pisces cumber line = Touched half way between the upper and lower surface of the aerofoil. Indicates the curvature of the wings.

The maximum thickness point = the solid part of the wings expressed as a percentage of the wire by changing each of the above attributes of aerofoil, the designer is able to adjust the feather's performance so that it is suitable for special work. For example, a crop duster may have a thick, high waist wing that produces large amounts of lift at a low speed. Alternatively, a jet has a thin wing with at least a camber, allowing it to cruise at high speeds.

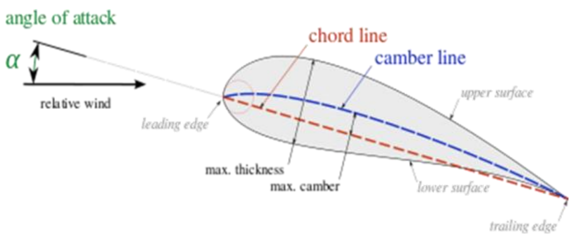


Fig. 3. Aerofoil diagram

B. How it works

The basic principle behind aerofoil is described by the Bernoulli's theorem. Actually, it says that the total pressure is equal to the constant pressure (due to the weight of the air) and dynamic pressure (due to wind speed). The air that goes to the upper surface of the aerofoil has to travel fast and thus the dynamic pressure increases. The loss of post-static pressure leads to a pressure difference between the upper and lower surfaces, which is called the elevator and opposes the weight of an aircraft (or pressure opposing the drag). As the angle of attack (angle between wire line and relative air flow) increases, more lift is made. Once the critical angle of invasion (usually around 14 degrees) reaches the aerofoil's stall.

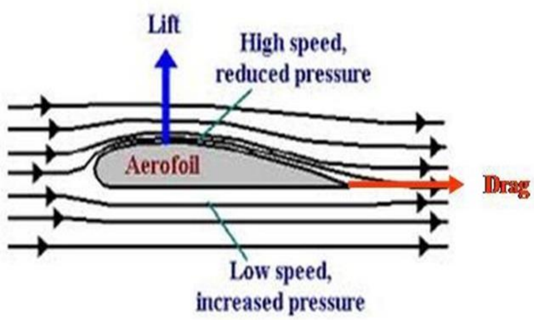


Fig. 4. Lift & drag working

C. Aerofoil Geometry

The atmospheric geometry may be characterized by the upper andlower surface coordinates. It is often summarized by some parameters such as maximum thickness, maximum blanket, maximum thickness position, maximum camber position and nasal radius. According to these parameters a proper air force section can be generated by Eastman Yakub in the early days of 1930 so that it can be known as NACA section. NACA 4 digits and 5-digit airfoil were made by super imaging simple meaning shapes with a thick distribution, which was obtained by making some popular aerofoil suitable for the time being

$$y = (t/0.2) \times (.2969 \times x^{0.5} - .126 \times x - .3537 \times x^2 + .2843 \times x^3 - .1015 \times x^4)$$

The 4-digit sections of the candle line were defined as a parabola for the maximum camber position from the leading edge, then another parabola back to the rear side. After the 4-digit class, there was a 5-point class like the famous NACA 23012. These classes had the same thickness distribution, but used a candle line with more curvature near the nose. For 5-digit sections, a cubic was fired in a straight line.

The 6 series of NACA aerofoil has left this simple defined family. These segments were produced by more or less determined pressure distribution and some laminas were used to obtain flow. After six-chain sections, the aerofoil l design became more special for a special application. Air force with good transnational display, good maximum lift capacity, very thick square, very few drag class is now designed for each use, often a feather design starts with the definition of several airflow sections, and then the entire geometry It is modified on the basis of 3-dimensional characteristics.

D. Ways of Aerofoil Design

The process of aerofoil design is achieved by the relation between boundary layer properties and the relationship between geometry and pressure distribution. The goal of the air film design is different, some aerofoil are designed to reduce (and the lift may not be required to generate all). Some parts need to be pulled down while producing a certain amount of lift. In some cases, the drag really does not matter - this is the maximum lift which is important for this performance, thickness, or pitching moment or off-design. This performance may need to be achieved with hindrances on opulence or other abnormal obstacles. Some of these have been discussed further in the section of the previous section of historical examples.

One way to aerofoil design is to use an airfoil which was already designed by a person who knew what he was doing. This "design by authorization" works well when the goals of a specific design problem match with the targets of the original AF design. This case is rarely low, though sometimes existing air flosses are quite good. In these cases, airplanes can be selected from catalos such as Wing section, Althaus 'and Wartman's Stuttgart profiles, Elthos' Low Reynolds Number Air flight Catalo, or Selig's "Airlines at Low Speed" as the principle of Abbott and Von Donhoff. The advantage of this approach is that test data is available. There is no wonder, as the unexpected early stall, the possibility is, on the other hand, the available tools are now adequately refined that one can surely be sure that anticipated performance can be achieved. Specifically, the use of "designer airflies" is very common in the requirements of a given project, this section of notes is related to the process of custom airfilm design.

E. NACA 0024 Airfoil Section

NACA 0024 aerofoil NACA is one of four-digit wing section series. It is a 21% thick, symmetric ammunition with a maximum thickness point located at X / C = 0.30, where X is the axial coordinate which is measured with the windline nose and the length of the wire is A complete description of airfoil can be found in Abbott and von Donhoff (1959). Figure 3 Figure 3 shows the NACA 0024 profile. The experimental data to be compared with CFD calculations were taken from Gregorick et. al. (1989). Was tested in an NACA 0024 airfoil section

The pressure of the Triassic air tunnel surface was measured with 56 faucets distributed with the surface and Reynolds number = 1.5×106. Only static-state data is used for the following comparison.

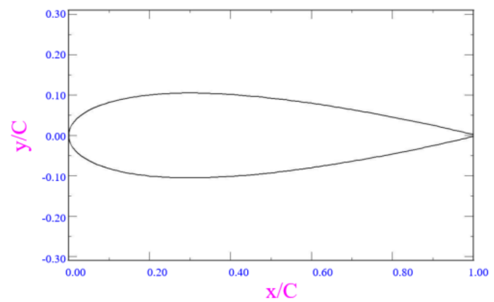


Fig. 5. NACA 0024 section profile

4. Computational fluid dynamic (CFD)

For the use of computational fluid dynamics (CFD), mathematics, physics and computational software, imagine how gas or liquid flow affects the gas or the fluid as well as it flows from the past. Computational fluid dynamics are based on Navy- Stokes equations. These equations show how the velocity, pressure, temperature and density of moving fluid are related. Computational fluid mobility has been around since the 20th century and many people are familiar with the form of an instrument for analyzing the flow of air around cars and planes. Since the complexity of the cooling infrastructure of server rooms has increased, CFD has also become a useful tool in the data centre for analyzing thermal properties and modeling air flow. The CFD software requires information about the size, content and layout of the data center. It uses this information to create a 3D mathematical model on a grid that can be rotated and viewed from different angles. CFD modeling can help an administrator identify the hot spots and helps to know why cold air is wasted or air mix. Just by changing variables, the administrator can imagine how many cold air data flow through the center in many different situations. This knowledge can help the administrator to optimize the efficiency of the existing cooling infrastructure and can estimate the effectiveness of a particular layout of IT equipment. For example, if an administrator wanted to take a rack of hard drive storage and wanted to split the hard drive into two racks, then the CFD program could simulate the change and helps the administrator understand that the extra heat What adjustments will be required to handle at any time or money before the expense of the money is loaded.

A. Why used computational fluid dynamics (CFD)

There are some important application explain in following.

1. Numerical simulation of fluid flow (enabled).
2. Architect to create a comfortable and secure environment.
3. Designer of vehicles to improve aerodynamic characteristics.
4. Chemical engineers to maximize yield from their devices.
5. Petroleum engineer to prepare optimum oil recovery strategy.
6. Surgeon to cure arterial diseases (computational hemo dynamics).
7. Forecasting weather forecasting and warning of natural disasters.
8. Safety experts to reduce health risks from radiation and other hazards.
9. Assessing military organizations to develop arms and damage.
10. CFD practitioners for making large bucks by selling colorful pictures.

B. CFD analysis process

- Information about problem statement about flow
- Mathematical model iBVP = PdE + IC + BC
- Mesh generation nodes / infinite volume cells, at times
- Space dissected coupled OD / DAE system
- Time Dissection Algebraic System X= B
- Effective Solver Discrete Function Value
- Implementation of CFD Software, Debugging
- Simulation run criteria, stop criteria

C. Description of the problem

- What is known about the problem of flow?
- Should physical incidents be kept in mind?
- What is the geometry of domains and operating conditions?
- Are there any internal barriers or free surfaces / interfaces?
- What is the type of flow (laminar / turbulence, stable / unstable)?
- What is the purpose of CFD analysis?

D. Mathematical model

1. Choose a proper flow model and the reference frame.
2. Identify the forces that cause the cause and effect of the fluid motion.
3. Define the computational domain in which to solve the problem.
4. Prepare conservation legislation for mass, speed and energy Simplify governing equations to reduce computational effort:
   - Use available information about the prevailing flow mechanism.
   - Check for symmetries and major flow directions (1D / 2D).
   - Ignore the results of which have little or no effect on the results.
   - Model of small-scale fluctuation effect that cannot be captured.
   - Including a primary knowledge (measurement

data, CFD result).

5. Add numerical relationships and specify the initial/ boundary conditions.

E. Dissection process

1. PDE system turns into a set of algebraic equations.
   - Mesh generation (decomposition in cells / elements)
   - Structured or unstructured, triangular or quadrilateral?
   - CAD tools + grid generator (Daylight, Forwarding Front)
   - Adaptive refinement in trap shape, 'interesting' flow areas.
2. Space deviation (estimation of spatial derivatives)
   - Limited Inter / Version / Element
   - High-versus low order estimation
3. Time deviation (approximation of temporary derivatives)
   - Clear versus implied plans, lack of stability
   - Local time-step, adaptive time step control
4. Coupled non-linear algebraic equations should be redone.
   - External recurrence: Discrete problem is updated using the coefficient.
5. Solution values from previous walkers.
6. Get rid of non-letters by a method like Newton
7. Solve governing equations in a different fashion
   - Internal iterative: the result is usually the result of linear sub problems
8. Solved by a recurrence method (conjugated gradient, multi gridge)
9. Direct solvers (Gaussian eradication) are prohibitive costly
   - Convergence Criteria: It is necessary to check residual, relative solutions.
10. Variations and other indicators to ensure that iterations are united.

In the form of a rule, the algebraic method is too big to solve (millions unknown) But sparse, that is, most matrix coefficients are equal to zero.

F. How to do CFD simulation

- Compute time for flow simulation depends on time.
- Choice of numerical algorithms and data structures.
- Prevent criteria for linear algebra tools, repeat solvers.

G. Discretization parameters

Some parameters are used in CFD like as mesh quality, mesh size, time step.

- Cost per year for convergent iterations and convergence rate.
- Programming language (most CFD codes are written in Fortran)
- Many other things (hardware, vector, parallel, etc.)

Simulation depends on the quality of the result.

- Mathematical models and implicit assumptions.
- Approximation type, stability of the numerical plan.
- Traps, time steps, error indicators, stop criteria.

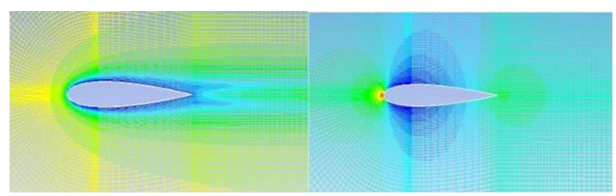
H. Post processing and Analysis

Simulation results are done in post processing order Remove the desired information from the calculated flow area

- Calculation of derivative quantities (streaming, vorticity)
- Calculation of integral parameters (lift, drag, total mass)
- Visualization (represented as number of pictures)
  - 1D data: function values are directly connected to the straight line.
  - 2D data: stylize, contour level, color diagram
  - 3D data: cutlets, cut planes, isonephs, ice walls
  - Arrow plot, particle tracing, animation. Systematic data analysis through statistical tools.
- Debugging, verification and validation of the CFD model

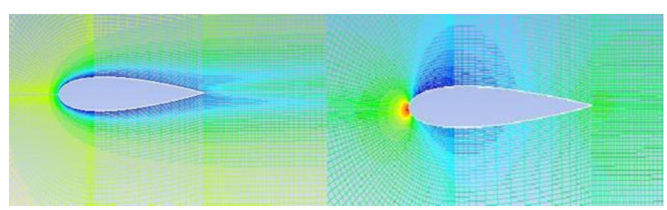
5. Results

Results are obtained in the form of pressure distribution and turbulence by using FLUENT 6.3.26 pressure distribution and turbulence are show in figure at leading edge, across the section and tailing edge. the variation of pressure distributions are also show at upper chamber and lower chamber of aerofoils. there are 4 Colors show in figure (blue, green, yellow and red) all colour are represented the compare of numerical values of the pressure distribution and turbulence. Blue, green, yellow and red these arrange in increasing order of numerical values of pressure and turbulence respectively. we can see variation of pressure distribution and turbulence by increasing angle of attack.



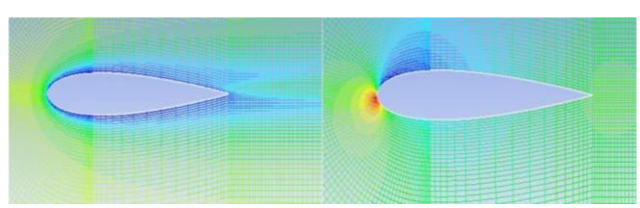
(a) (b)

Fig. 6. Pressure and Turbulence distribution at 0° angle of attack



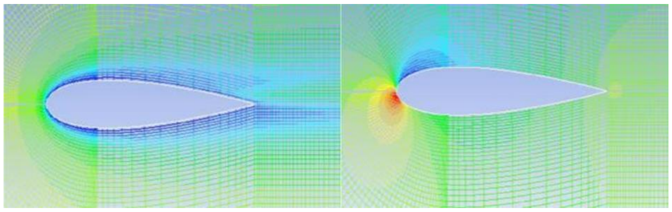
(a) (b)

Fig. 7. Pressure and Turbulence distribution at 2° angle of attack

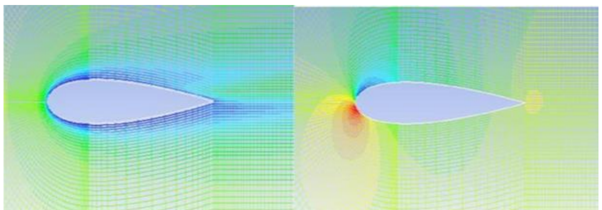


(a) (b)

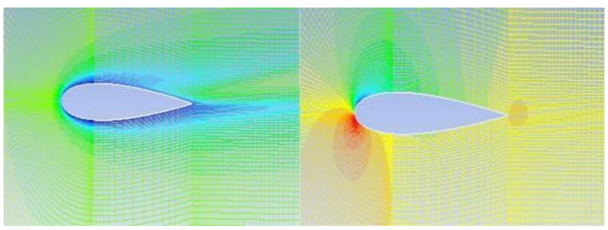
Fig. 8. Pressure and Turbulence distribution at 4° angle of attack



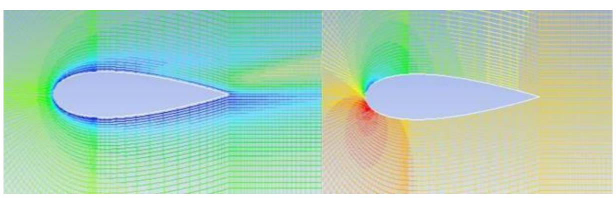
(a) (b)
Fig. 9. Pressure and Turbulence distribution at 6° angle of attack



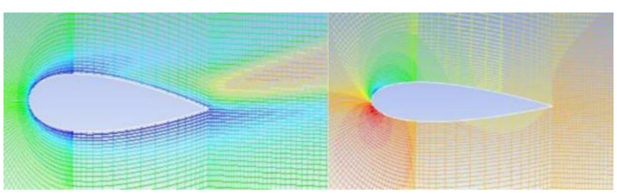
(a) (b)
Fig. 10. Pressure and Turbulence distribution at 8° angle of attack



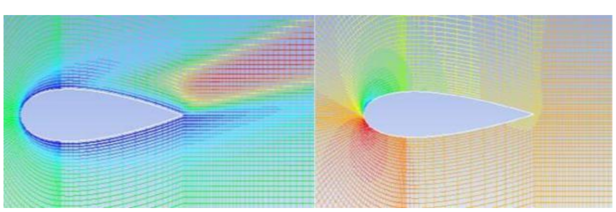
(a) (b)
Fig. 11. Pressure and Turbulence distribution at 10° angle of attack



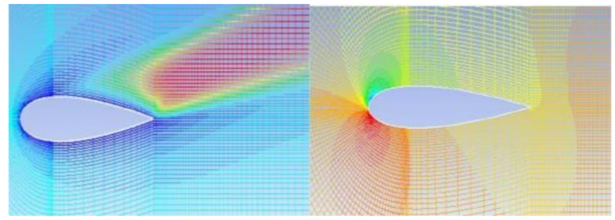
(a) (b)
Fig. 12. Pressure and Turbulence distribution at 12° angle of attack



(a) (b)
Fig. 13. Pressure and Turbulence distribution at 14° angle of attack



(a) (b)
Fig. 14. Pressure and Turbulence distribution at 16° angle of attack



(a) (b)
Fig. 15. Pressure and Turbulence distribution at 18° angle of attack

A. Discussion

Lift and drag coefficient were measured experimentally in wind tunnel tests for NACA 0024 airfoil. Lift and drag coefficient were mainly influenced by increasing angle of attack and decreasing. If the angle of attack increases, then lifting up to a certain angle and increasing drag coefficient. After some angle, while the lift coefficient was decreasing; And drag coefficient was increased. This situation was called as the Stoll Angle, the stove angle transitioned due to unrest flow from Laminar. In addition, lift and drag coefficient were calculated with CFD analysis, which was used for Spar tart Lamar and KE-Epsilon. Spar tart Alma, K-Epsilon and experimental results were compared. Two methods are compared with experimental results. Numerical Solution Method Is Better than K-Epsilon In addition to the 4-A and 4-B illustrated in the picture as equality experimental results, lift and drag coefficient were given in Fig. 6. The best result for lift and drag coefficient was achieved by between 10° -14° attacking angles. In addition, the best performance of the NACA 0024 (CI / CD) was shown in the picture 5. Evaluation

The result of the numerical simulation of the flow characteristics around the aerofoils section of the axial flow fan is compared to ten different parameters of the angle of attack as 0° 2°, 4°, 6°, 8°, 10°, 12°, 14°, 16° and 18°. To get the full picture of the flow around the blade section, the variance variations of different forms are compared with pressure distribution and unrest, and these can help better blade designs and fans in particular.

The pressure distribution at nine places of aerofoil around the aerofoil are obtained in the aerofoil and at the leading edge, to simulate the amount and direction of air flowing around the aerofoil. pressure distribution shows that the pressure distribution intensity around and around the aerofoil is changing. Limitation separation is not seen in simulations and there is pressure variation in the aerofoil because the angle of the attack is comparable to the leading and the leading edge for the low or 0° value. As the angle of attack is increased to 0° to 18°, the pressure distribution shows the increasing value of pressure in the upper part of the section and the decreasing values of pressure in the lower part. The pressure distribution on the back side also show the same pattern. With angles of the cover of 0 °to 18°at 2° alternate pressure, it is obtained to show the difference in the pressure around the aerofoil. As pressure is shown, pressure distribution on the leading edge is rapidly getting different as the angle of attack is increasing. The value of the pressure on the upper side of the aerofoil section is lower than the lower part of the aerofoil section. With the increase of the angle of attack, the pressure distribution around the aerofoil is becoming the same on the trailing edge, the optimum value of the attack angle is less than 14 ° and greater than 10° for the maximum values of the lift and drag ratio.

The forms obtained from the numerical simulation have also been compared and important information about the unrest has been given. Figures show that the flow for the low and 2 value of the angle of attack is very smooth. As the angle of attack is increasing, the turbulence in the lower part of the blade section is increasing. The flow on the upper side is very smooth The maximum storm looks backwards for an attack angle of 10° to 14° and it is the maximum near the edge leading to the angle of 10 ° to 14° attack. The optimum value of the disturbance can also be found in the middle of the angle of 10 ° -14 ° for the smooth flow of air around the aerofoil.

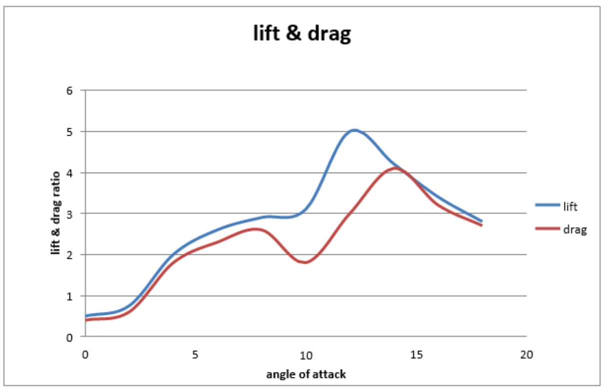


Fig. 16. Lift and drag chart

6. Conclusion

The demonstration and drag of the NACA 0024 airfoil was demonstrated in this study. Numerical and experimental results were compared to a flight program used for numerical calculation. The results of the calculation were given as follows: Drag and raise the hike with the rising angle of the attack. The stall was started with an attack angle of 10° to 14°. Lift coefficient while decreasing; Drag coefficient increased. The optimum lift coefficient value was measured and calculated at 10° to 14°.

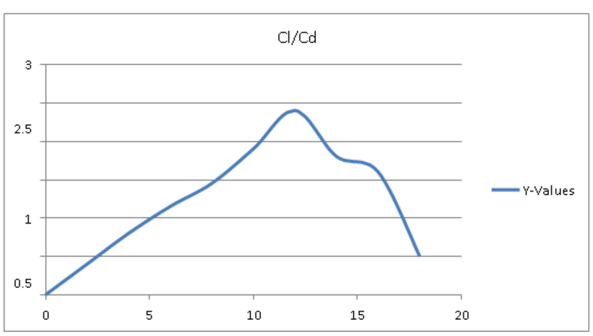


Fig. 17. Lift and drag ratio

7. Scope and future work

In the present distribution work, the numerical study of the aerofoil section of the blade of axial flow fan has been studied. Simulation results are presented in the form of pressure distribution and turbulence in the form of angle of attack as 0° 2°, 4°,6°,8°,10°,12°,14°,16° and 18° for 10 values of angle of attack. In the standard form the numerical unrest CFD is used for analysis.

The following can also be done in the future

1. Numerical analysis can be done on other section of aerofoil.
2. Other unrest models can also be used for other sections of aerofoil.
3. Experimental analysis can also be done for numerical study done in the work of this dissertation.

References

[1] Durga Sanskrit Pangharhi and Devi Prasad Mishra (2014). CFD simulation for selection of suitable blade profiles for improving energy efficiency in axial flow mine ventilation fans. Journal of Sustainable Mining, 13 (1), 15-21.

[2] Ahmed F. Abdel Azim El-Saeed and Mostafa Mohammad M. Ibrahim, "Numerical Investigation of Different Tip Clearance Shaped Effects, Performance on A Axel Flow Compressor Stage", The Online Journal on Power and Energy Engineering (OJEPE).

[3] LE Yang and Elu JE, Journal of Hydrodynamics Science Direction, "Internal flow mechanism and experiments with squared blade in front of low pressure axial fan".

[4] Hussain Norie, Florent Ravetlet, Fareed Becker, and Christophe Saaraf, "Experimental Detection Dictated Counter-Rotating Axial Flow Fans", Author Manuscript, "Proceedings of ASME-JSME-KSME Combined Liquid Engineering Conference 2011, published in Hamamatsu Shizuka" Japan (2011). "

[5] c. Surf, H. Nauri, F. Revlet, F. Goat, "Manuscript published in Proceedings," Experimental Study of Blade Thickness Effects on a Comprehensive and Local Performance of a Controlled Warnet "ASEE-JSME-KSME Combined Liquid Engineering Engineering Hail - 00507786, Version 2 - 25 November 2010.

[6] Eck, B., "Fans", Pergamon Press, 1973.

[7] Ali Akturk and CengizCamci, "Platform Platform Extensions Using Axle Flow Fan Tip Leakage Flow Control", Journal of Fluides Engineering, May 2010, Vol. 132/05110 9 -1.8

[8] Laszlo Malnar, Janos Wad, "The design of small axial flow fan rotor of high specific performance",

[9] D. N. Rensen, Mcc Thompson and J. N. Sorensen, "Improved Rotor-Only Axel Fans-Part II: Design Optimization for Maximum Efficiency", Journal of Fluids Engineering June 2000, Vol. 122, 325.

[10] Imad e Ehhadi and Wu Kaki, "Numerical Simulation and Amendment in the Axial Flow Fans, Events of 3D Flow", Works Quarter 7 No. 2 (2003), 199-213.

[11] Yug-kyun Kim, Tae-gu, Jin-Heukhar, Sung-Jaya Moon, and Jay-Hein Lee, "Floorantistics Impactor Change an Axle Turbo Fan", International Journal of Aerospace and Mechanical Engineering 4: 1 2010.