CFD Study on Turbulence and Boundary Layer Flow Modelling for External Aerodynamics

Prashant D[#1], Chandru B T[#2], Dr. Maruthi B H[#3],Ravindra A R[#4].
Mechanical Engineering Department, EWIT, Bengaluru-91
KARNATAKA, INDIA
[#1] P G STUDENT,pprashi108@gmail.com, [#2] ASSOCIATE PROFESSOR,[#3] PROFESSOR & HOD.
[#4]P G STUDENT,ravindra.0905@gmail.com

***Abstract*— This project shows the performance of several steady Reynolds-averaged Navier-Stokes (RANS) turbulence models and boundary-layer modelling approaches is evaluated for an airfoil, by comparison with empirical data for a Reynolds number ranging up to 9 million. In this work two dimensional CFD analyses have been carried out to predict the aerodynamic forces on the surface of the NACA 63(2)415 airfoil. The shear stress transport (SST) k-ω turbulence model performs exceptionally well when combined with boundary layer modelling approach, which shows that the turbulence model characteristics are particularly suitable to deal with this specific flow problem. This project focus on knowing the accuracy of the results obtained with different modelling techniques employed for turbulence and boundary layers. RANS equations are time averaged equations for fluid flow motions .Finally CFD analysis results are compared with the wind tunnel data available in the literature survey. The deviation between CFD analysis results and wind tunnel test data are founded out and accurate procedure for modelling the near wall flow region and especially about the stalling region in the airfoil has been concentrated.**

***Keywords*— Reynolds number, boundary layer,deviation.**

Introduction

Aerodynamics is the branch of study of forces which is when the body is considered while it is moving through air. These strengths are created because of the relative movement between the air and the surfaces of the article.

Airfoil

The reason for this segment is to talk about the connection between airfoil geometry and airfoil execution. An airfoil-shaped body develops a differential pressure distribution across its surface with the lower surface having a larger pressure compare d to upper surface .This differential pressure is the origin of the lift acting on the airfoil. The perpendicular motion produced due to this is call lift. Flight airfoils have a trademark shape with a balanced driving edge, trailed by a sharp trailing edge, consistently withlopsided camber.

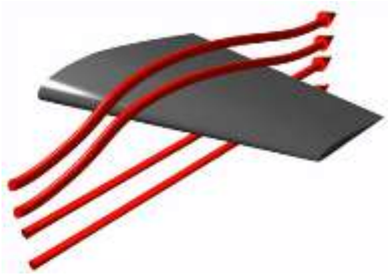


Fig 1.1: Airfoil Section

Because of the shape of an airfoil and its respective angle of attack the lift depends and also on its camber. By Bernoulli's guideline for incompressible liquids in viscid stream the speed distinction will happen, which thusly creates the lift power.

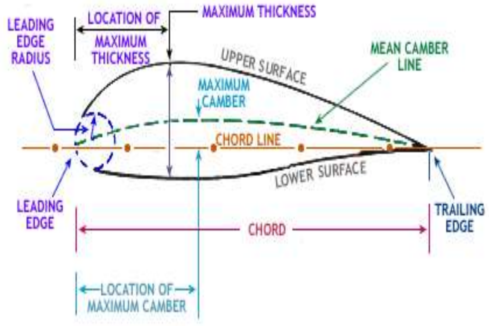


Fig 1.2: Terminology of an airfoil

The different terms identified with airfoils are characterized beneath:

- The suction surface is by and large considered with lower static weight and higher speed.
- The weight surface has higher static weight than the suction surface. The weight distinction between these two surfaces contributes the lift in an airfoil.
- The agreement line which is a straight line interfaces the principle and trailing edges, at the terminations of the camber line.
- The harmony is the distance of the harmony line and is the trademark measurement of the segment.
- The most extreme size and the area of greatest size are communicated as a rate of the harmony.

- The streamlined focus is the distance about which the pitching minute is autonomous of the approach and lift.
- The focal point of weight is the harmony area about which the pitch minute is zero.

A Nozzle to be a device designed as for as restraint the characteristics of a fluid flow (especially as for as increase velocity) as it exits an enclosed cavity via an orifice.

A nozzle is frequently a pipe or tube from make cross sectional area and it can occur used as for as direct or modify the flow from a fluid (liquid or gas). They are frequently used as for as restraint (control) the rate from flow, speed, direction, mass, shape, and or the pressure from the stream.

The nozzle aim is to increase the kinetic energy of the flowing medium towards the expense from its pressure and internal energy. Nozzles can occur delineate while convergent or divergent (expanding from a smaller diameter as for as a larger one). A de Laval nozzle is a convergent section followed from a divergent section and to be often called a C-D nozzle. C nozzles speed subsonic fluids. If the nozzle pressure ratio is more sufficient the flow will extend sonic velocity towards the narrowest point (i.e. the nozzle throat). In this condition, the nozzle is said to occur choked. Convergent divergent nozzles can therefore accelerate fluids that possess choked in the convergent section as for as supersonic speeds. This C-D process is greater efficient than allowing a convergent nozzle to expand supersonically out ward. The divergent section also certain that the direction from the escaping gases is backwards and sideways trying not contribute to thrust.

Propulsion system is a system fundamental obeys Newton's laws that force is proportional as for as rate of change of momentum, and that action and equal. Thrust chamber is one example system that works using these laws besides some time ago systems that is turbojet and ramjet. Cavity is by expelling stored subject, called the propellant. This thrust can range from mega-Newton to mille-Newton. Thrust chamber system can be main spacecraft propulsion that is missile launcher, assist-take-off engines for airplanes and even ejection of crew escape capsules.

OBJECTIVES

- The objectives is to increase thrust
- Analysis of flow through C-D Nozzle for different altitudes
- Obtained maximum uniformly of flow by achieving linear thrust
- To achieve linear velocity which will lead to maximum thrust
- To prove CFD is an effective tool for analysis

Forces on Airfoil

Point of impact is the particular point at which the air separated to flow about the airfoil. Thus the aggregate streamlined power is coupled when wind currents over and under an airfoil, which thus make the airfoil to apply by the push.

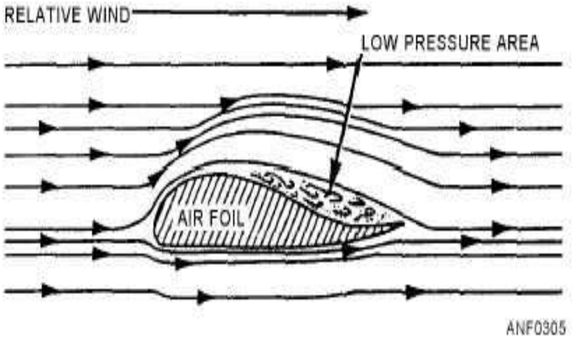


Fig 1.3: Flow around an airfoil

The stagnation point is framed at the purpose of effect which is considered as a high weight region. It (high-weight territory) is situated at the lower part of the main edge, contingent upon approach. This high-weight zone adds to the general power delivered by the cutting edge.

Figure exhibits wind current lines that diagram how the air moves about the airfoil range. By third law which expresses "each activity has an equivalent and inverse response." Since the air is being redirected descending, an equivalent and inverse power must act upward. This power adds to the aggregate streamlined power created by the airfoil. At low approaches, the redirection compel or affect weight may apply a zero positive constrain, or even a descending or negative power.

Aerodynamic forces will be produced in other way that air is passing over the top of the airfoil. As per Bernoulli's Principle the airfoil causes a low weight over the airfoil, and the reduction in weight on top applies an upward streamlined power. Indeed, even a little weight difference produces significant power when connected to the huge territory of the sharp edge rotor.

The two segments called lift and drag is because of the aggregate streamlined power, might be isolated into two segments called lift and drag. The power follows up on the airfoil in a heading opposite to the wind is considered as Lift. Drag acts parallel to the relative wind. Numerous elements changes the lift (complete) delivered by an airfoil. On the off chance that the rate expands, it causes the expanded lift on the grounds that a bigger weight distinction is been delivered between the upper and lower surface of an airfoil. Lift dependably does not increment in direct extent to speed, but rather changes as the square of the rate. Along these lines, an edge going at 1000 bunches has four times the lift of the same sharp edge going at just 500 bunches. Likewise Lift fluctuates with the zone of the sharp edge segment. The region of 1000 square feet will create twice as much lift as a cutting edge range of just 500 square feet. Approach considerably affects the lift delivered. Lift increments as the approach increments up to the slowing down approach. Slow down edge fluctuates with various edges and is the time when wind current no more takes after the camber of the cutting edge easily. Air thickness is another element that specifically impacts lift.

Lift

Lift is portrayed to be the section of the force that is inverse to the drawing nearer or free stream heading.

Streamlined force is the one where the fluid is thought to be air. Streamlined lift is most regularly associated with the wing of an adjusted wing aircraft, lift is can likewise be created by rotors if there should arise an occurrence of helicopter, sails, wings on auto dashing autos and so on. Yet, the regular importance of "lift" demonstrates that lift restricts gravity, lift can be in any bearing.

Non-streamlined questions, for example, feign bodies and plates (not parallel to the stream) may likewise produce lift while moving in respect to the liquid. This lift might be consistent, or it might waver because of vortex destroying. Communication of the article's adaptability with the vortex shedding may improve the impacts.

Lift Coefficient

The lift created for particular stream conditions can be resolved utilizing the accompanying condition where the predefined approach is known:

$$Cl = \frac{L}{\frac{1}{2}\rho V^2 S_{ref}}$$

Where
L -lift force, and C_L-Coefficient of lift
P-air density
V-true airspeed
A-plan form area

Drag

In liquid flow, drag alludes to powers that restrict the relative motion of an article through a fluid. Drag qualities depend on upon pace and other resistive forces, for instance, dry crushing, which is free of pace. The segment opposite to this bearing is considered as lift. In this way drag restricts the movement of the item. In optimal design, climatic drag can be viewed as a wastefulness requiring cost of extra vitality amid dispatch of the space object or as a reward streamlining come back from circle.

Drag Coefficient

The drag coefficient of any framework in an air contains the impacts that two fundamental givers can be given to liquid element drag, skin rubbing and shape drag. The drag coefficient of a lifting airfoil likewise incorporates the impacts of affected drag.

The drag coefficient *Cd* is defined as:

$$C_D = \frac{2F_d}{\rho \vartheta^2 A}$$

1.1.1 Bernoulli principle

In the eighteenth Century, the Swiss mathematician Mr. Daniel Bernoulli found that the weight of a liquid declines at point where the pace of the liquid increments. This implies the rapid stream makes low weight and low speed stream makes high weight. As the particles of air act simply like atoms of liquid this rule applies to air. On account of an airfoil an atom hitting the main edge will accelerate as it goes over the extensive bended surface, this will make a low-weight region on the grounds that the particles are further separated. The particle that hits the main edge and goes underneath will travel slower as it has less separation to cover, which implies the atoms, will be denser. Both upper and lower atoms will meet in the meantime at the trailing edge.

1.1.2 Flow regimes

The Mach number is characterized as the proportion of stream pace V, to the velocity of sound an in a medium. It demonstrates the significance of the compressibility of the liquid. This proportion is critical in the liquid stream issues since weight unsettling influences engender in a liquid at the nearby speed of sound and the compressibility of a liquid allows a sound wave to travel. The rate of sound in a liquid is identified with the route, in which thickness and weight shift.

Stalling region of airfoil

Stall is a marvel in which the air ship wings create an expanded air resistance and diminished lift, which may bring about an airplane to crash. At the point when the wind current isolates from the upper wing surface thus the slow down happens. It happens when a plane is under excessively extraordinary an Angle of Attack For light airplane, without high-lift gadgets, the basic edge is generally around 16°.

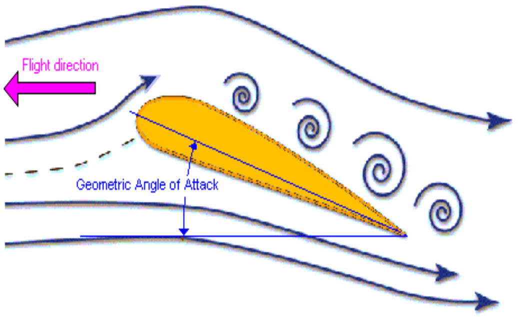


Fig 1.4: The stalled airfoil

A stall is a condition considered in streamlined features wherein the approach increments past a specific point such that the lift continues diminish step by step. The basic approach is the one in which edge happens now. This basic edge is needy upon the profile of the wing, its arrangement structure, and different elements, yet is ordinarily related with the approaching wind for most airfoil that shifts in the scope of 8 to 20 degrees.

Methodology

Airfoil selected: NACA 63(2)415

Reynolds number considered for flow simulation: 9 million Experimental (wind tunnel test results) are available for Reynolds number of 9 million in references, Abbott.I.H,'Theory of wing section, including a summary of airfoil data",

Stage1. Structured mesh is generated using commercial meshing tool ICEMCFD.

Stage2. CFD analysis is carried out in FLUENT CFD software with two different boundary layer modelling techniques.

CFD analysis is carried out using wall function approach (k-epsilon model),

Stage3. CFD investigation results are contrasted and the test (wind burrow test results) results and conclusions are arrived.

Assumptions

The stream around the airfoil is dealt with as enduring, Compressible.

3.3 Governing Equations

The Governing Equations of CFD are the surely understood "Navier-Stokes" conditions. They portray the preservation of mass, force and vitality of a streaming liquid.

In 3-Dimensions we can write the compressible Navier-Stokes equations

$$\frac{\partial U}{\partial t}+\frac{\partial G_1}{\partial x}+\frac{\partial G_2}{\partial y}+\frac{\partial G_3}{\partial z}=\frac{\partial G_{1v}}{\partial x}+\frac{\partial G_{2y}}{\partial y}+\frac{\partial G_{3v}}{\partial z}$$

Where,

U is a vector of preserved variables, given by $U=\begin{bmatrix}\rho\\ \rho u_1\\ \rho u_2\\ \rho u_3\end{bmatrix}$

G1,G2 and G3 are the In viscid Flux vectors,

$$G1=\begin{bmatrix}\rho u_1\\ P+\rho u_1^2\\ \rho u_1 u_2\\ \rho u_1 u_3\\ P u_1+\rho u_1 E\end{bmatrix}\qquad G2=\begin{bmatrix}\rho u_2\\ \rho u_2 u_1\\ P+\rho u_2^2\\ \rho u_2 u_3\\ P u_2+\rho u_2 E\end{bmatrix}$$

$$G3=\begin{bmatrix}\rho u_3\\ \rho u_3 u_1\\ \rho u_3 u_2\\ P+\rho u_3^2\\ P u_3+\rho u_3 E\end{bmatrix}$$

G1*v*, G2*v* and G3*v* are the viscous flux vectors given by

$$G1v=\begin{bmatrix}0\\ \tau_{xx}\\ \tau_{xy}\\ \tau_{xz}\\ u_1\tau_{xx}+u_2\tau_{xy}+u_3\tau_{xz}-q_1\end{bmatrix}$$

$$G2v=\begin{bmatrix}0\\ \tau_{yx}\\ \tau_{yy}\\ \tau_{yz}\\ u_1\tau_{yx}+u_2\tau_{yy}+u_3\tau_{yz}-q_2\end{bmatrix}$$

$$G3v=\begin{bmatrix}0\\ \tau_{zx}\\ \tau_{zy}\\ \tau_{zz}\\ u_1\tau_{zx}+u_2\tau_{zy}+u_3\tau_{zz}-q3\end{bmatrix}$$

Where,

P is the Pressure (Pa)

E is the Total Energy, Sum of Internal Energy & Kinetic Energy.

$$E = e + \frac{1}{2}(u_1^2 + u_2^2 + u_3^2)$$

The internal Energy e is given by,

$$e = \frac{P}{\rho(\gamma - 1)}$$

γ is the ratio of Specific heats

$$\gamma = \frac{C_p}{C_v}$$

The Pressure, Temperature and density are related by the equation of state as,

$$P = \rho RT$$

Where, R is the Gas Constant

T is the Temperature

The internal energy e is evaluated as follows,

$$e = C_v . T$$

$$\frac{P}{\rho} = RT$$

$$R = C_p - C_v$$

$$R = Cv(\frac{C_p}{C_v} - 1)$$

$$\frac{C_p}{C_v} = \gamma$$

$$R = C_v(\gamma - 1)$$

$$R = C_v(\gamma - 1)$$

$$C_v = \frac{R}{\gamma - 1}$$

$$e = C_v . T = \frac{R.T}{\gamma - 1}$$

$$e = \frac{P}{\rho(\gamma - 1)}$$

Where μ is the dynamic viscosity coefficient

Heat Fluxes

$q1$,$q2$ and $q3$ are the conduction heat fluxes, given by,

$q1 = -k\partial T/\partial x$

$q2 = -k\partial T/\partial y$

$q3 = -k\partial T/\partial z$

K is the Thermal conductivity in $wm{-}k$

In viscid approximation is the situation in which viscous and heat conduction effects are neglected.

The In viscid approximation is valid in large regions of the flows around the bodies, except the regions close to the solid-surfaces where boundary layer effects are important.

The Inviscid compressible Navier-Stokes equations are called **"Euler Equations"**, obtained by Ignoring the viscous terms of Navier-Stokes equations.

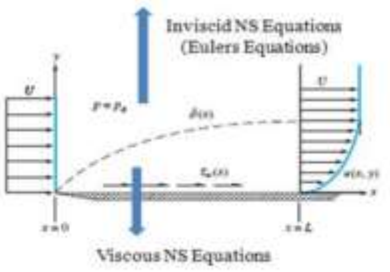


Fig 3.1: Applicability of Viscous and Inviscid NS equations

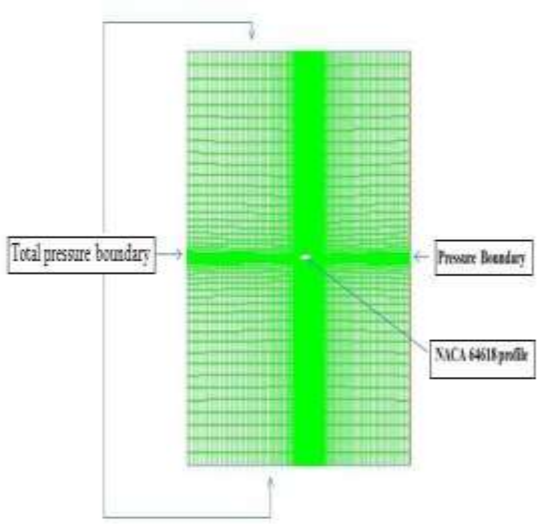
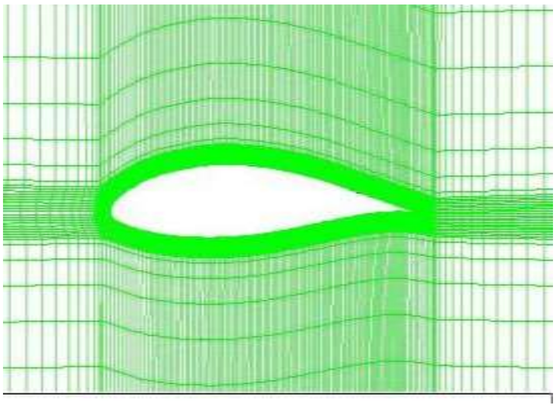


Fig 3.6: Computational geometry for 2D CFD analysis



RESULTS AND DISCUSSIONS

The table 4.1 shows the wind tunnel test (Experimental) Results, contains the Lift and Drag Coefficients for every two degree angle of attack from 00-200. These outcomes are taken from the books "Hypothesis of wing areas" by Abott, The diagrams of this book is connected in the supplement.

Table4.1: Wind tunnel test (Experimental) results for NACA63(2)415 airfoil at Re=9 million.

Angle Of Attack (Degrees)	**Lift Coefficient (Cl)**	**Drag Coefficient (Cd)**
0	0.4	0.0048
2	0.6	0.005
4	0.9	0.0053
6	1.1	0.0086
8	1.28	0.0132
10	1.43	0.0148
12	1.5	0.015
14	1.55	0.0165
16 (Stalling)	1.6	0.017
18	1.58	0.018
20	1.56	0.02

Above Table shows the wind tunnel test results for NACA 63(2)415 airfoil at Reynolds number of 9 million, corresponding to Mach number of 0.38, which is in the compressible flow regime. The maximum lift coefficient at stalling attack is 1.6 and the corresponding drag coefficient is 0.017.

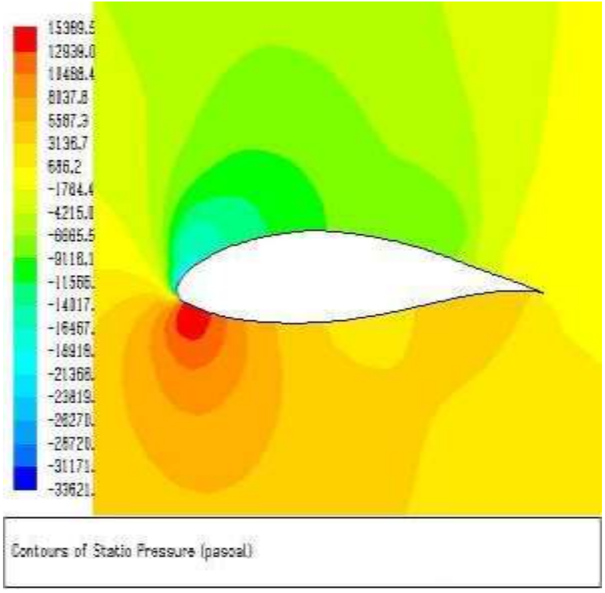


Fig4.7: Pressure vector at 180 AOA (Post-Stall) by SST Model

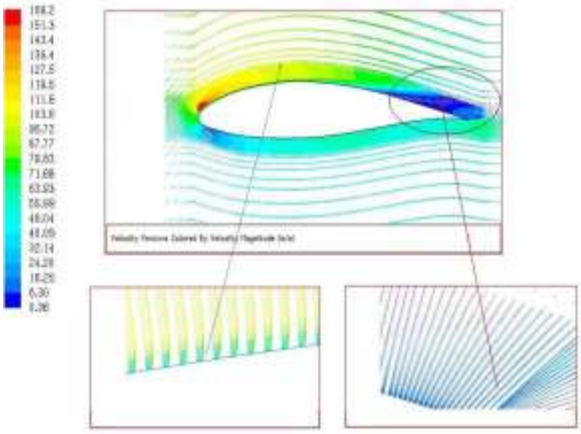


Fig4.8: Speed vector at 180 AOA (Post-Stall) by Mentors SST turbulence model

CONCLUSION

From the above talks the lift and drag coefficients taken from the test wind tunnel test results are 1.6 and 0.017 independently at a methodology 160 (Stalling Region).similarly 1.48 and 0.0159 are Lift and Drag coefficients gained by the K-Ɛ Model, 1.59 and 0.0165 are Lift and Drag Coefficients got by the Mentors SST turbulence model. It is watched that lift and drag coefficients processed with Mentor's shear stress model are near the trial results measured at the wind burrow. It is watched that the stream division locale is very much caught by the Mentors shear stress model and thus the Mentors shear stress model is effectively anticipating the lift coefficient in the slowed down district.

Subsequently it is inferred that Mentors shear stress transport Turbulence model utilizing the immediate determination of limit layer area by a fine work gives the exact CFD examination results in the pre slow down and in addition post slow down district

REFERENCES

1. Abbott.I.H,'Theory of wing section, including a summary of airfoil data", Dover book on Physics, 1995
2. David Hartwanger et.al "3 D modeling of a Wind Turbine using CFD"NAFEMS Conference, United Kingdom, 2008
3. Frank Bertagnolio et.al "Wind Turbine airfoil catalogue"RISOE National Laboratories, Denmark, 2001
4. H.Gao et.al "Computational study of unsteady flows around dragonfly and smooth airfoils at low Reynolds number"46th AIAA Aerospace sciences meeting and exhibit,Reno,Navada,2008
5. J.E.Bardiana et.al "Turbulence modeling validation study"NASA Technical Memorandum 110446, 1997
6. Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," AIAA Journal, Vol. 32, No. 8, August 1994, pp. 1598-1605 (http://turbmodels.larc.nasa.gov/sst.html)
7. D.C.Wilcox,'Turbulence modelling for CFD'DCW Industries, California, 1993
8. Anderson DA., Tannehill JC. andPletcher, RH. "Computational Fluid Mechanics and Heat Transfer, " Hemisphere Publishing Corporation, McGraw-Hill Book Company, 1984
9. Anderson John D., "Computational Fluid Dynamics", McGraw-Hill, 1995. 12. John Anderson, Jr.," Introduction to Flight", McGraw –Hill, 2000 13. F.M.White,"Fluid Mechanics", McGraw-Hill, 2005