CFD Analysis of NACA 4412 Aero Foil with Flap

R.Praveshkumar[1], A.Somaraju[2]
[1]Dept of Mechanical Engineering
[2]Assistant Professor, Dept of Mechanical Engineering
[1, 2] D.J.R College of Engineering and Technology.

Abstract- *In this project work, an aerodynamic investigation of NACA4412 aerofoil wing with flap is carried out numericallywith the help of ANSYS FLUENT ® software.The aerodynamic characteristics of the aerofoil namely lift;drag and moment coefficients are determined turbulent flow by considering SST k-ω turbulencemodel***.***Reynolds Number is varied as*$1x10^6$*,*$2x10^6$*,*$3x10^6$*,*$5x10^6$*,*$7x10^6$*,*$10x10^6$ *for different angles of attack* 0^0*,*5^0*,*10^0*,*15^0*and* 20^0 *respectively.The aerodynamic charecteristics are obtained in the form of lift curve,drag curve and momemtum curve. Pressure velocity and flow vectors are also obtained. It is observed that as Reynolds number increases aerodynamic performance coefficient like lift and drag coefficients increases and moment coefficient decreases*

Keywords- CFD,angle of attack, aerofoil, Reynolds number and aerodynamic performance.

I. INTRODUCTION

1.1 INTRODUCTION1

An,aerofoil is the shape of a wing. A body in the shape of an airplane moved1through a fluid produces an3aerodynamic force.1The component of0this force8perpendicular to the direction of motion is called lift. The component1parallel to direction1of motion is called drag. The lift of an aerofoil is primarily the result of angle of attack and shape (in particular its chamber). When either of the two is positive, the resulting flow field around the aerofoil has a higher average velocity on the upper surface than the lower surface.This difference in speed is accompanied by a pressure difference (according to the Bernoulli principle) which in turn produces a lift force. The lifting force can also be related to the upper / lower average speed difference without invoking the pressure using the traffic concept. The wings of a1fixed.wing,horizontal1and;vertical4aircraft built0with cross1sections in the form of anaerodynamic profile. swimmers and flying creatures are also use the aerodynamic profiles .Any object that has an2angle1of attack in the air or a moving6fluid1such as a1flat plate will generate an aerodynamic force called lift. Aerofoils are more efficient forms of liftingcapable of generating more lift (up to a point) with less drag. They must be the most basic and fundamental characteristics to take into account when designing an aerodynamic profile for a flight of wing.

1.2 NACA AEROFOIL

The aerodynamic wings of NACA are forms created1by. The0National Aeronautical Advisory7Committee,(NACA). The1shape3of2the NACA aerodynamic profilesis represented by a digit progression that follows "NACA". The2parameters0in the1numerical7code`can`be put in conditions to1create2the cross3section of the1wing aerofoil3.

1.3 NACAaerofoil profile geometry

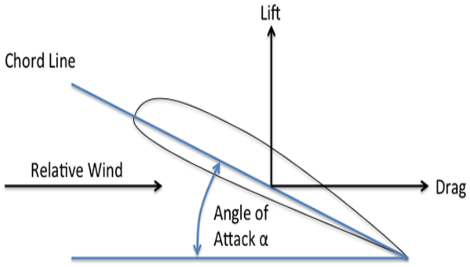

Fig.1.1 - Schematic representation of the aerofoil

The figure 1.1 shows the schematic diagram of aerofoil of aeroplane wing with representations of the relative wind velocity and angle of attack with the forces acting on it as liftand drag coefficient. The force,acting`parallel to;the aerofoil direction is *drag force*, the`force which is perpendicular to the aerofoil is called *lift force*.

II. LITERATURE-REVIEW

2.1. Introduction

Current-project is based`on aerodynamic analysis1of NACA4412 with flap.the main objectives of this project is to determine and identify the drag,lift,momentum,pressure velocity and flow vectors at several angles attack and Reynolds number.

Triet*et. al.*[1]have carried out experimental tests in wind tunnel where the flow situations are modelled after actual flight conditions.Reliable results can be obtained by dimensional analysis. The cross section of a typical airplane wing is an aerofoil and is largely responsible for producing the forces that sustain the aircraft in flight.

Zhang Y*et.al.* [2] have tested an advancement outline technique for supercritical common laminar stream aerofoil in view of hereditary algorithm and computational fluid dynamics. Class shape change technique is received as geometry parameterization. Limitations on weight circulation are connected to increase suitable stream field. It is agood compromise for supercritical natural laminar flow aerofoil in the design condition.

Parashar [3]has conducted the study to obtain the lift and drag coefficients for flow over NACA+2415, profilewith attack of angles from -15° to/+15°. He also plotted lift and drag curves from lift and drag coefficients.It was discovered that the shapes of the aerofoils have pronounced effect on their aerodynamic characteristics as they behaved differently in the same flow fields.

Al-kayiem*et.al*.[4] have stated thatduring ground aerodynamic collision characteristics-of the-aircraft wingchanges so much that the stream field structure around the flying body is bothered because of the impedance with the ground.

They also studied anaerodynamic behaviour of a NACA^4412 aerofoil underground collision. Reynolds number varies from 1.0 x 10^5 to 4.0 x 10^5 and angles of attack from -4°to20°. It was found out that the aerodynamic characteristics were strongly influenced at angles of attack of 4° to 8°.

Recktenwald0[5] has tested^{1}a2circular0plane form body non3spinning with6an aerofoil section^{3}in the6Auburn University9wind tunnel4facility. The aerofoil configuration was developed and produced by1Geobat Flying Saucer Aviation Inc. For6comparison purpose, a1Cessna0172 model8was also5tested. The1author found0that slope of the lift6curve of4for Geobat1model was^{0}less than1that of3Cessna0172 but4displayed better4stall4characteristics.

Ravi et.al. [6] predicted numerically a transition model of an incompressible laminar to turbulent flow over NACA 4412 air foil at Reynolds number of 3.0 x 10^6. The-lift and drag+coefficients are obtainedfrom this study, using the SpalartAllmaras and Shear Stress Transport (SST) turbulence models with transition capabilities, were compared with experimental data and concluded that the SST turbulent model had better results in both the pre-stall and post-stall region on the computational model.

Eleni*et.al*. [7] studies on the variation of coefficients drag and lift from flow around NACA(0012 aerofoil of Reynolds number 3.0 x 10^6 for different viscous turbulence models such as the Spalart-Allmaras, the realizable k-ε and the SST models. They concluded that SST had better agreement with experimental data

Dwivedi*et.al.* [8] have embraced a straightforward approach for investigate streamlined static steadiness examination of various sorts shapes of wings.he tried the decreased scale estimate wings-of various shapes like-rectangular, rectangular.with bended tip, decreased, decreased with bended tip, and so on in low speed subsonic breeze burrow at various velocities and distinctive approaches. The creators1found0that the decreased1wing0with bended2tip2was2the most1stable1at various speeds and scopes1of working1angles2of1attack approaches.

Mineck1***et.al.***[9]have tried^{1t}hree1planar, untwisted1wings with1the same circular harmony1yet1with various shapes1of the1quarter-harmony1line. They1found that1the circular wing1with the upswept1quarter-harmony line1has1the most minimal1lifting productivity,1the1curved1wing1with1the upswept1trailing1edge has1the most2astounding lifting proficiency2and1the sickle formed1wing has1effectiveness ..

Qin*et.al*[10] have numerically studied the DGE of a delta wing and found that with decreasing ride1height, the1lift, drag2and1nose-down^{2}pitching2moment increased nonlinearly, and1the increment1became larger2with increase in sink1velocity. In case of 2D airfoil, Nuhait [11] and Chen [12] numerically simulated the landing1process0of a?2D flat plate and found that the significant changes in lift occurred due to DGE.

Qu.*et.al.* [13] have tested the DGE of NACA4412 aerofoil by solving the RANS equations with SST)k-ω0turbulence model. The flow field data including weight, speed and streamlines was exhibited in detail, and the pressure work impact was acquainted with clarify1why the2lift in2DGE is bigger than2that in1SGE.

R.Wahidi1.***et.al.*** [14]have researched to impacts of laminar partition rises by1measuring over0NACA4412 aerofoil demonstrate utilizing volumetric1three segment1velocimetry (V3V)3and molecule picture velocimetry1(PIV) at2Reynolds number050000 and0diverse approach. They characterized that beginning of change, area of division and1reattachment after2they made1remark connections of various6Reynolds typical and1shear pressure by means of the time found the middle value of V3V. Their outcomes showed that move up of span wise caused4vortices and3these vortices roused1sets of1negative1and1positive divider ordinary speed. The1sets completed a tremendous demonstration in4the vulnerability of0the1reattachment of5the isolated shear2layer.

M. Agrawal*1et.al* [15] have pointed the investigation for the streamlined of wings. They chose NACA 4412 aerofoil model and accomplished couple of comes about subsequent to finishing wind burrow tests segments. They saw from comes about that lift expanded until a specific point if approach expanded. Their perception additionally incorporated that drag began to end up noticeably a prevailing variable if approach continued rising and slow down happened because of every one of these parts.

Alex E.ockfen.*et.al.*[16] . have tested The turbulent stream around an aerofoil in outrageous ground}impact with flap was considered in PC^{1}program$^-$Fluent,0a limited volume1code utilizing the isolated1SIMPLE1solver. The conditions administering0the 2D-:incompressible0stream are the conditions of RANS , with Spalart-Allmaras-turbulence demonstrate actualized to give conclusion. With the expansion of a fold in extraordinary-ground closeness the impacts of fold avoidance, ground-tallness, fold sort, approach, &Re, considered. outcomes give knowledge of streamlined qualities in the stream, and in addition useful flight administrations for ground nearness flight with a flap.

Omar Kemal Kinaci*et.al* [17] have stated that an iterative limit component strategy for an answer of the stream around a wing-in-ground impact was proposed. Contrasted with RANSE arrangements BEM is very quicker; which just discretizes the body inside the liquid as opposed to the entire liquid space. Be that as it may, as appeared in this paper, there are techniques to expand the effectiveness of BEM. Iterative BEM is speedier and has great exactness, along these lines; it is by all accounts more productive than coordinate BEM. Iterative strategy indicates great prospect in managing geometries in ground closeness where a higher number of boards must be utilized. These may incorporate complex collections of three measurements in which case the boards will never again be spoken to by lines yet surfaces which will build the computation time fundamentally. Brisk evaluation of the effectiveness of a specific kind of wing segment for WIG artworks can be made with IBEM. At the point when the wing is in outrageous ground impact, the wing will conceivably waver noticeable all around because of vortex partitions from the body. This will make precarious movements for the wing. Shaky movements will lead the WIG issue to be explained at each time step expanding the need for snappier arrangements. It is felt that IBEM will demonstrate its genuine worth in flimsy situations where the arrangement time is relied upon to drop altogether. As a future work, the produced code will be created for understanding shaky movements to incorporate outrageous ground impact cases.

Shaowei Lia,***et.al*** [18]Have prove that the streamlined attributes of aerofoil RAE2822 at various ground clearances and attack angles is completed. It appears: Realizable k-epsilon show is extensively shown with great capacity of foreseeing the stream attributes. In investigation of the ground impact of RAE2822, high lift and proportion of lift and drag just could be accomplished in medium approach. The stationary ground condition would incite the detachment of limit layer on the ground and impact the streamlined trademark when the airfoil is near the ground. The compressibility of flying demonstrations a noteworthy effect on the streamlined qualities of airfoil in ground impact, it ought to be considered in numerical recreation and test.

M. Nazmul Haquea.*et.al* [19] Have stated the regions from the centre^{1}wingtowards .the1root^{1}increments and1towards1the tip1the region diminishes in.a similar1rate.Generally speaking-surface1zone of2the1wing stays same1as1the-rectangular plan form. Thus, the wing can deliver more lift because of expanded surface zone close to the root. In the meantime, stream partition along the traverse of the wing is lessened because of slow diminishment of harmony length along the traverse thus the drag is likewise lessened. From the examination of exploratory information it´is observed that coefficient of lift is bended at driving edge plan form increments and the coefficient of drag diminishes at approaches beneath 12˚ in contrast with the rectangular plan form; though basic point of assault does not shift altogether between the two plan forms. Past basic approach, estimations of lift and drag coefficients are practically equivalent. In that capacity, it can be inferred that the bended driving edge plan form displays preferred streamlined execution over the rectangular plan form because of higher lift to drag proportion at approaches beneath the basic approach.

III. METHODOLOGY

3.1 .Introduction

The-general methodology of.the Computational Fluid dynamic software fluent and the modeling software solid works. It explains about the steps to be followed in the projectwork to obtain the solution.

3.2 Computational Methodology

Computationalstudy ofaerodynamic1flow1over the NACA 4412 aerofoil with a chord length of 1.0 m was studied numerically in this project work. simulations are done for attack anglevaries between 0° to 20^0. The density, ρ of the1air is taken as 1.225 kg/m^3 duringcaculation. The Reynolds numbers considered in this study ranged from 1.0 x 10^6 to 10.0 x 10^6. Within this range, the flow can be considered as incompressible for simplicity. The computational domain was created using ANSYS Design Modeller while the unstructured meshes with quadrilateral elements were created in pre-processor ANSYS FLUENT. The resolution and density of the mesh are higher inthe near wall region of the aerofoil. The generated mesh had a total of 252726 nodes and 125864 elements.

In general, the process of problem is defined in 5 simple step.

1) 1Geometry
2) 2Mesh
3) 3Setup
4) 4Solution
5) 5Results

3.2.1*Model Preparation

profiles were acquired in the form of coordinated vertices, that is, documents written1and4imported in the5ANSYS programming. Some0minor?conformities have been made4to correct the4geometry and make1it substantial1as a CFD0model.

ANSYS`is essential in the time dedicated to the CFD exam: it creates a work environment where the thing is duplicated. A basic part of this is to make the cross section, including the article. This must be connected in all directions to obtain the physical properties of the wrapping liquid, in this case, moving air. The work and edges must also be assembled remembering the true purpose of characterizing the basic concept

3.2.2Meshing

A computational mesh of the NACA aerofoil is created by using one semi circle geometry and 2 squares. To get the good accurate results,fine mesh is created near the aerofoil area and coarse mesh is generated for for away areas from the aerofoil.An1organized quadratic grid was used for meshing the aerofoil.The lattice must be fine in like manner in particular regions far away1fromthe2aerofoil. A1fine1job deducts2a greater number1of accounts,1which^{3}makes the1simulation^{1}is long1forNACA-aerofoils, the1especially front has1an1edge organize passed on with1a growing partition1between centers, starting1from little3sizes constrain conditions6satisfactorily.

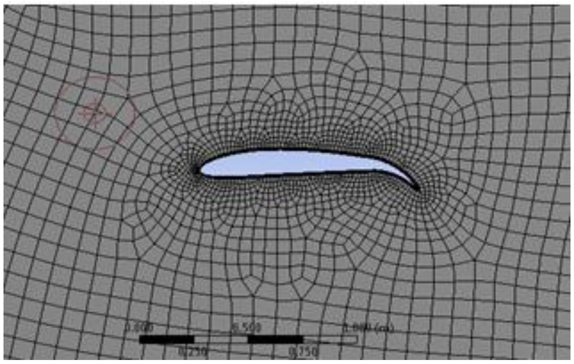

Fig.3.1 Computational domain with mesh

The figure 3.1represents the discretized model ofaerofoil in the computational domain. The computational domain is the region where fluid flows from inlet to outlet. In this project work, the fine mesh is generated for aerofoil with flap. For this, the region around the aerofoil is inflated in order that the fluid has a smooth and well distributed pathway. Skewness is calculated to determine how good the quality of the mesh is. Skewness is ranged from 0 to1 where the mesh with elements closure to the value 1 is a of bad quality. One can observe in the below figure that most of the elements are in a good quality region. Here the mesh settings have set as good quality mesh settings.

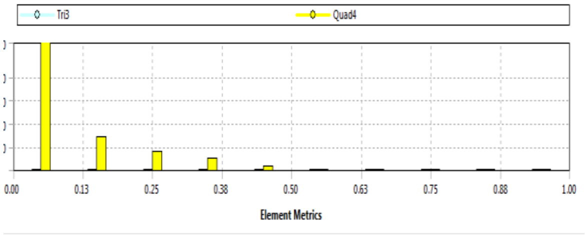

Fig.3.2. Skewness representation of the elements

3.2.3 Boundary condition setting

Giving the properties to the unmistakable geometries is basic for amusement of the model. Proper boundary conditions are imposed on the simulation model to get the more accurate results. TheX and Y0components0of air4velocity is set as1inlet velocity4and at the9outlet of the aerofoil, the property "pressure” is set as zero to produce the zero gauge pressure.

Boundary Conditions

The imposed boundary conditions, which include inlet, outlet and wall boundary conditions, are as indicated in Fig.3.3

Inlet: The velocity inlet boundary condition is set which assumes the average velocity of the flow as the inlet velocity. For non-zero angles of attack, the x and y components of the velocity are given by$U_x = U\cos\alpha$, $U_y = U\sin\alpha$

Outlet: This is the boundary condition set at the outlet domain. Theoutlet boundary condition is set as zero pressure, which assumes ambient atmospheric conditions.

Wall:slip boundary conditions are imposed. The upper and lower aerofoil surfaces are activated as wall boundaries with both velocity components set as zero value.

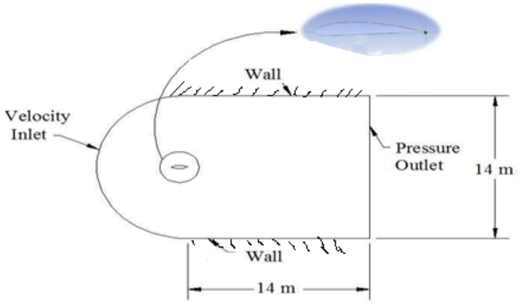

Figure 3.3Computational domain with boundary conditions

The figure 3.3 represents the schematic of the boundary conditions applied to the aerofoilwith flap for given computational domain. Inlet velocity is given usually according to the Reynolds number and the angle of attack considered. The following few lines give the ANSYS FLUENT.

Setting up Boundary Conditions in FLUENT:

The geometry was imported into FLUENT from SOLID WORKS and the mesh is created in FLUENT. The1environment properties and “Double1precision" is1chosen asframework parameters1guaranteeingsufficient exactness. FLUENT has single4exactness as default, however2for these5reproductions a precise arrangement is asked. The residuals4for the distinctive7turbulence model variables were set as 10-8 and the4cycle max5check as to01000(iterations) . The1procedure of simulation couldbe ceased when C_L or C_D appeared to have balancedout,appropriately.

In the above discussed processes, the entities from 1 to 3 will be defined as a *pre processing* where we define the problem, discretize geometry and setup the problem in pre-processing the volume extraction and designing Boolean operations and the Boundary conditions. Name selections is given like inlet, out let and wall boundary conditions.

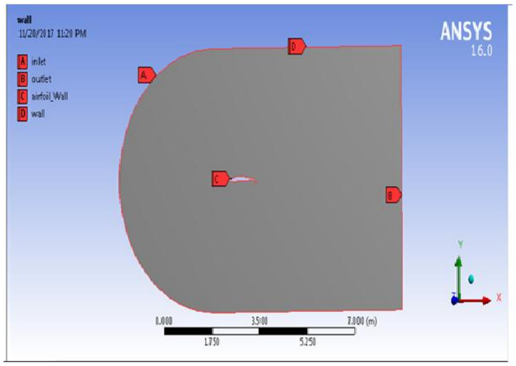

Figure 3 .4 Boundary conditions. And named selections

In the figure3.4, the named selections are indicated with alphabets like A-Inlet, B - Outlet, C-Aerofoil_wall, D-Wall to determine the easiest way possible to remember.

3.2.4 Solutionby Panel1Method

In this method well known courses for comprehending incompressible liquid stream more than 2D and 3D geometries. In 2D, the aerofoil surface©is for the most part partitioned into1straight1line sections or0boards. On1each-board are1put1sheets-ofvertex quality 'γ. Vortex’sheets are utilized in light of the fact that (smaller than expected vortices strength= γ0 s, s being length of a board) vortices offer ascent to flow, which thus deliver lift.

IV. RESULTS AND DISCUSSION

A finite volume based CFD code, ANSYS Fluent was used to solve the computational flow problem.Simulations were carried out for NACA4412 wing profile using SST turbulence model and panel method of solution. Air is consider as incompressibleflow for simplicity.

4.1Validation of simulation model

Before proceeding with the actual simulation work, one has to verify the results obtained from commercially available ANSYS FLUENT for correctness of results. For this purpose, the result obtained from fluent are compared with experimental values for known case of given initial and boundary conditions or can be compared with benchmark results. The verification can also be done against experimental data available in open literature. In this project work,the simulation results of NACA4412 aerofoil obtained from fluent are compared for validation purpose against the experimental data reported by Abbott and Doenhoff [21]under similar operating conditions. Results of lift coefficients drag coefficients, and momentum coefficients were obtained for different angle of attack for an aerofoil with Reynolds number of $6x10^6$ and turbulence effect. The present simulated resultsand experimental data are plotted and are shown in the figures 4.1, 4.2, 4.3 for comparison.

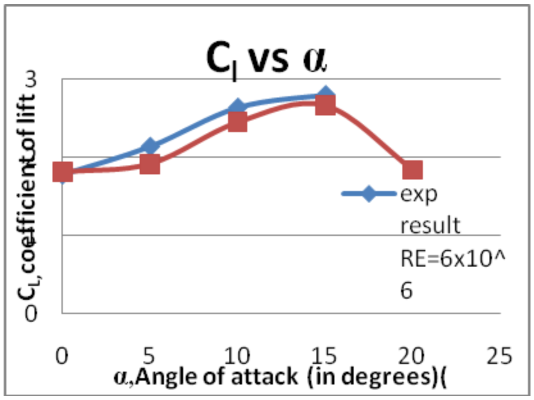

Fig 4.1.Graph of lift coefficient with experimental validation of Re =$6e^6$

It is seen from figure 4.1that coefficient of lift increases with increase in a attack angle for both experimental and numerical simulation work. CFD analysis give the results with higher values of lift coefficient compare to the experimental values, however the difference in results is about 10%.

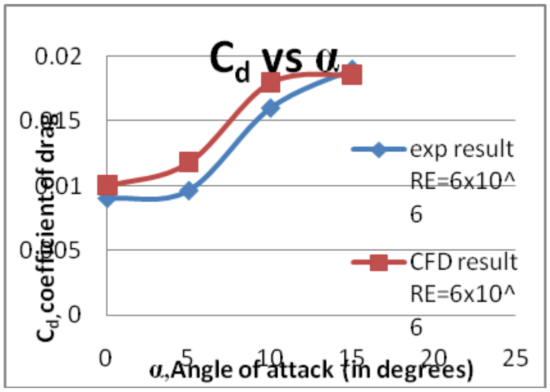

Fig4.2Graph of Drag coefficient with experimental validation of Re=$6e^6$

From the figure`4.2 represents the variation of the coefficientdrag withattack angle. It is observed from figurethat coefficient drag increases with increase in attack angle for wind tunnel experimental work and simulation work. CFD Simulation has over- predicted the values of drag coefficient as compared to the experimental values.

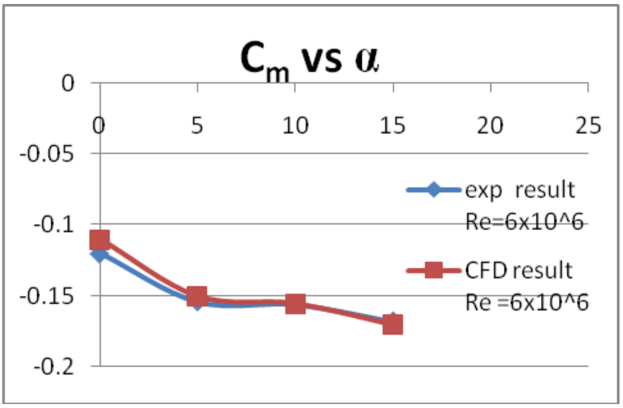

Fig 4.3 Graph of C_m with experimentalvalidation of Abbot with Re =$6e^6$

However the error in simulation results is 10%.The figure 4.4shows graph of variation of coefficient of momentum(C_m) with (α) attack angle where moment coefficient decreases as angle of attack increases. The comparison of experimental and numerical results show that simulated results are in$good correspondence within an error of 10%.

From the above three graphs it is clear that the results obtained from CFD simulation are in close agreement with experimental data from wind tunnel test and thus established a basis for the adoption of the computational method using FLUENT software.

4.2 Parametric Studies:

After validating the CFD model , varies simulations are conducted to evaluate aerodynamic operation of aircraft=wing aerofoil & to obtain the aerodynamic characteristics in terms of coefficients of lift drag and moment. In order to find the effect of a parameter of interest on the aerodynamic performance of aerofoil, the parameter of interest is varied while keeping all other variables as constant and this is known as parametric study.

In this project work, on two types of parametric studies are carried out where angle of attack is varied in one parameter keeping Reynolds number constant in one study and Reynolds number is varied in another study. The analysis has done for 0, 5,10,15,20 degreesof attack angle and for various Reynolds numbers of $1x10^6$,$2x10^6$,$3x10^6$,$5x10^6$,$7x10^6$ and $10x10^6$. The aerodynamic charecteristics are obtained in the form of lift curve, drag curve and momentum curve.ThePressure contours and Velocity vectors are also drawn to know the flow field and distribution in the computational domain.

Results of parameter studies are expressed in terms of contours and plots or graphs

4.2.1 Results in the form of contours:

Results for primitive variables like velocityand temperature are obtained by solving Navier Stoke equations. Pressure is obtained by solving pressure poissonequation .
In this project work , computations are carried out for isothermal conditions of the wing and hence no temperature contours are drawn. But only pressure ,velocity contours and velocity vector plots are drawn.

Case 1: Zero degree1angle of1attack

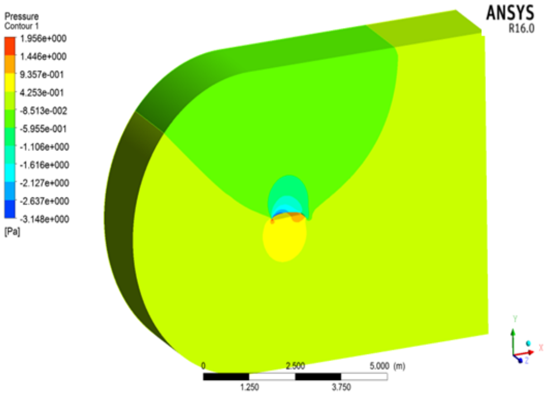

Fig 4.4 Pressure1contours at angle of1attack 0 degrees

The figure 4.4 Represents the Pressure contours of the Air around the volume of the aerofoil with flap where in the left picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum Pressure region

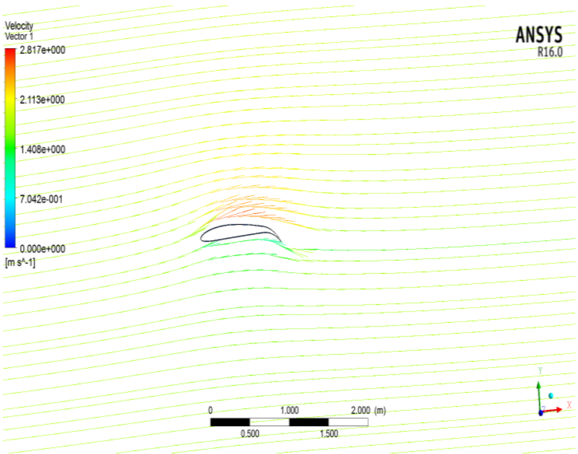

Fig 4.5.velocity Vector at an angle of attack 0 degrees

The figure 4.5 Represents the Velocity direction of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which isknown as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum velocity region

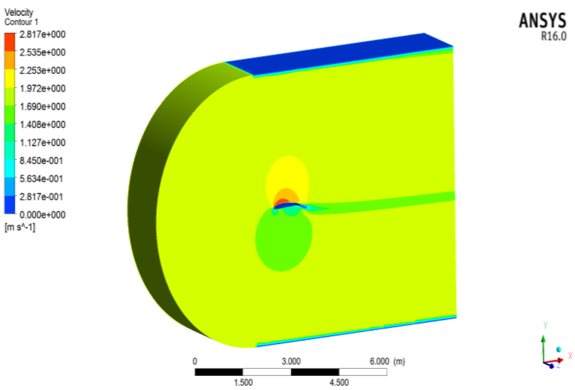

Figure 4.6Velocity contours atan angle 0 degree.

The figure 4.6 Represents the velocity contours of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum velocity region

Case 2: 5^{o}angle of attack

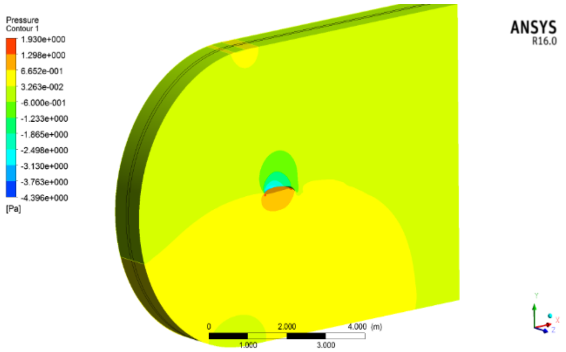

Fig 4.7 Pressure2contoursat angle of attack 5degree

The figure 4.7Represents the Pressure contours of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum Pressure region.

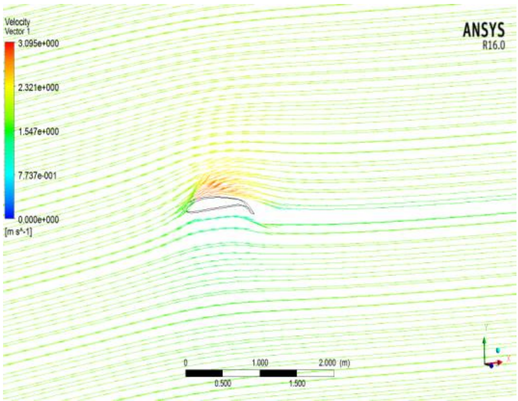

Fig 4.8. velocity Vector at an angle of attack 5 degrees

The figure4.8Represents the Velocity direction of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum velocity region.

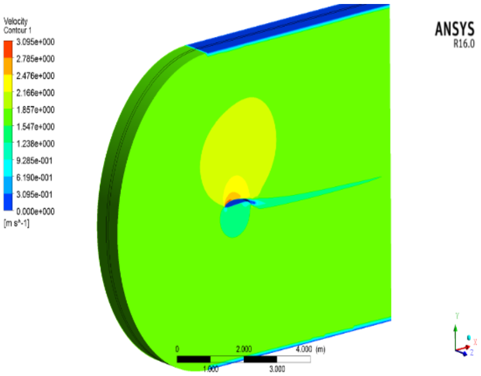

Figure 4.9.Velocity contour at an angle attack 5 degree.

The figure 4.9Represents the velocity distribution of the Air around the volume of the aerofoil with flap where in the left sidepicture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum velocity region.

Case 3: 10^0 angle-of-attack

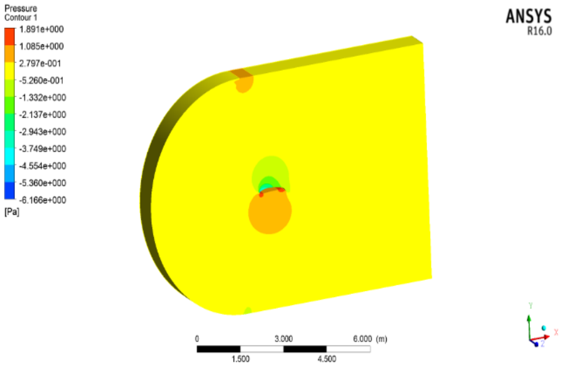

Fig 4.10 Pressurecontours1 at an .attack,angle10 degree.

The figure 4.10represents the pressure distribution of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum Pressure region

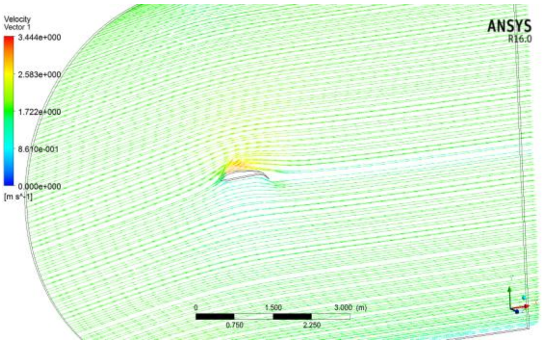

Fig 4.11.velocity Vector at an attack angle10 degree.

The figure 4.11Represents the Velocity direction of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum velocity region.

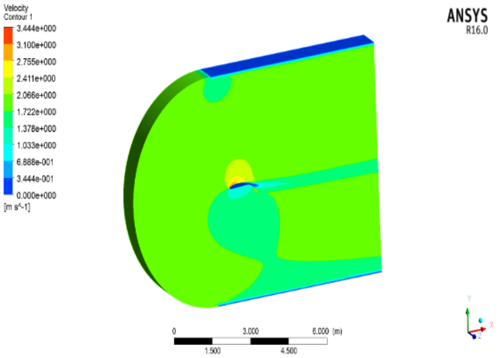

Figure 4.12. Velocity contoursat an attack angle 10 degrees.

The figure 4.12 Represents the velocity distribution of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum velocity region

Case 4: 15^0 angleθof.attack

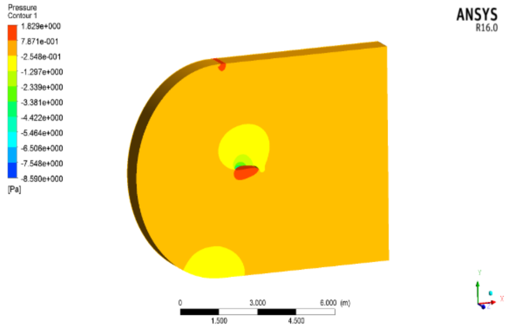

Fig 4.13. Pressure contours at an attack angle 15 degree

The figure 4.13Represents the Pressure distribution of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum Pressure region.

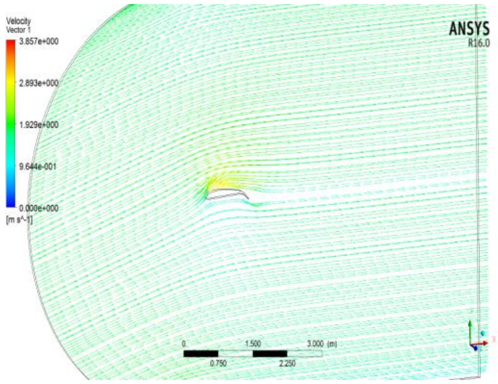

Fig 4.14. velocity vector at an angle of attack 15 degree.

The figure 4.14Represents the Velocity direction of the Air around the volume of the aer foil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum velocity region.

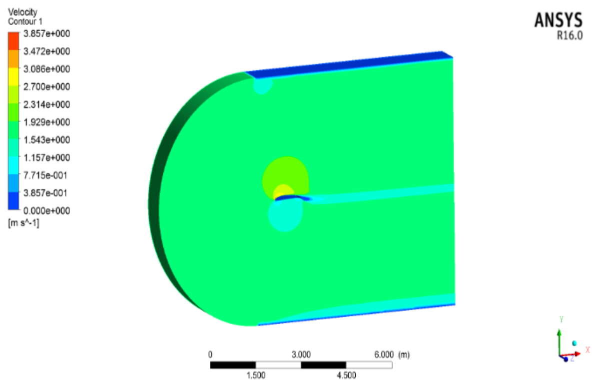

Fig4.15.Velocity contours at an angleαof attack (α)15 degree.

The figure 4.15 Represents the velocity distribution of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum velocity region.

Case 5: 20^oangleαof attack

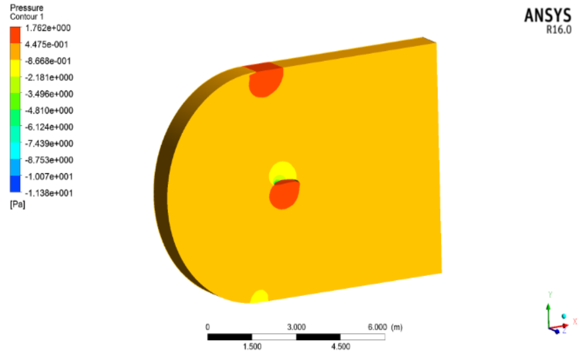

Fig 4.16. Pressure contoursatan angle of attack (α) 20 degree

The figure 4.16Represents the Pressure distribution of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum Pressure region.

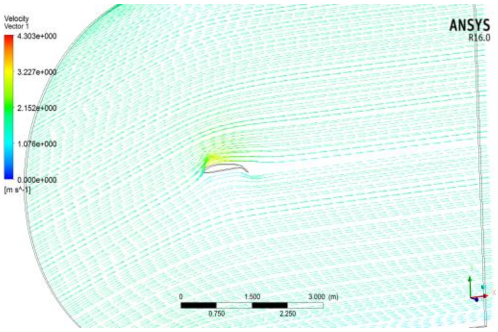

Fig 4.17.Velocity vector at an angle of attack 20 degree

The figure 4.17 Represents the Velocity vectors of the Air around the volume of the air foil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum velocity region.

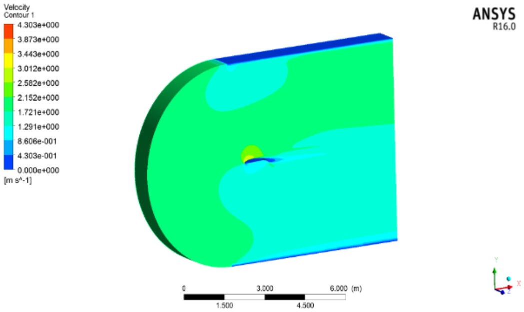

Figure 4.18.Velocity contours at an angle of attack 20 degree.

The figure 4.18 Represents the velocity distribution of the Air around the volume of the aerofoil with flap where in the left side picture is a coloured bar which is known as legend where the blue coloured region indicates minimum velocity region and the red coloured region indicates the maximum velocity region

4.2.2 Results in the form of graph

The aerodynamic characteristics of aircraft wing aerofoil mainly coefficients of lift, drag and moment are obtained from CFD simulation by varying angle of attack for varies Reynolds number and the results are plotted in the form of curves or graphs.

Case 1: lift curve

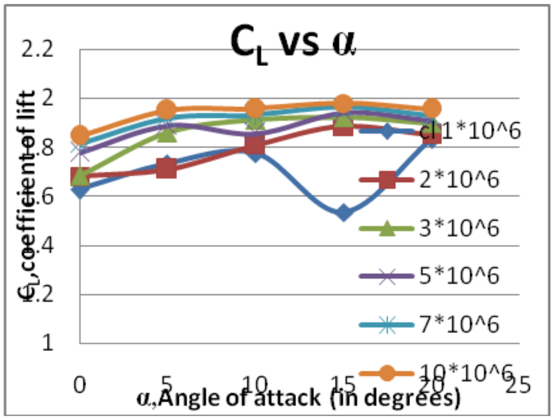

Figure 4.19C$_{L,}$(coefficient of lift) againstαangle of attack with Different Reynold numbers.

Thefigure 4.19shows variation of C_l with an attack angle at various Reynolds number.It is seen from figure that C_l (coefficient of lift) increases with.increase,inα (angle of attack) and Reynolds number.

Case 2 drag curve

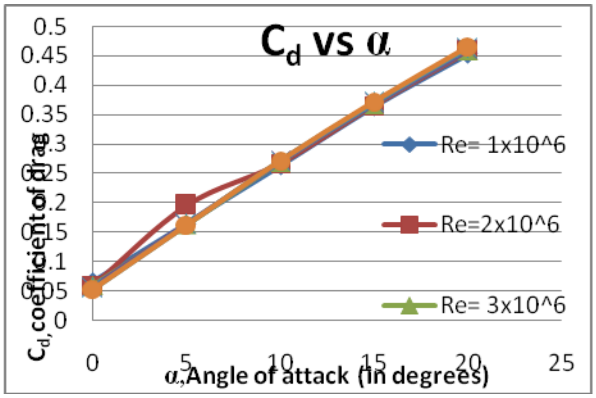

Fig 4.20. coefficient of drag(C_d)against angle6of8attack(α) with various Reynolds number

The above figure4.19 shows variation of $C_d$coefficient of drag with attack angle at varied Reynolds number.It is observed from figure that drags coefficient increases with increase in attack angle and Reynolds.number.

Case 3 moment curve

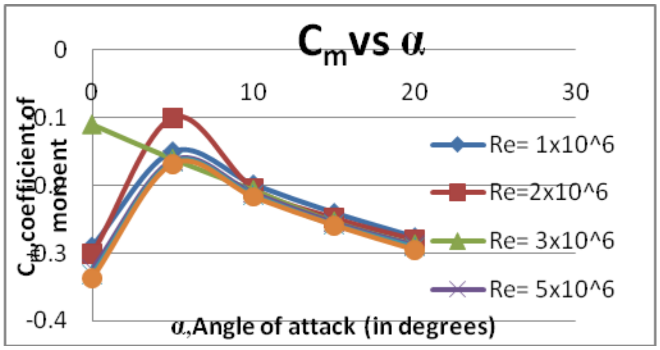

Fig4.21. Moment coefficient C_m againstα(angle of3attack)with Re (Reynolds)*numbers

The figure 4.21 display the variation for coefficient of momentum(C_m) with angle-of-attack at different Reynolds-number.as indicated,in the-figure, moment coefficient increases steadily for angles of attack between 0^0 to 5^0.beyond this angle a gradual decrease in moment coefficient is observed for different Reynolds number.

V. CONCLUSION AND SCOPE FOR FUTURE WORK

The numerical simulation of an aerodynamic performance of NACA 4412 aerofoil with flap at different angles of attack was studied using the SST k-ω turbulence model. The aerodynamic characteristics interms of coefficient lift ,drag and momentum are obtained from CFD simulation using ANSYS FLUENT software.

5.1. Conclusions:

results-of the simulation study carried out in project work.

1. The numerical computations were carried out by using ANSYS FLUENT.the model and the geometric designing is carried out in the SOLIDWORKS® CAD software.
2. Aerodynamic performance of aircraft wing aerofoil of NACA4412 profile is evaluated in terms of aerodynamic characteristics by using ANSYS FLUENT software. Aerodynamic characteristics are obtained for various values of Reynolds number by varying angle of attack 0^o ,5^o,10^o,15^o and 20^o.
3. As α (angle) varies the C_d (coefficient of lift) increases and thus shows improvement in aerodynamic performance.As the (α) increases 0^o to momentum)decreases.
4. As Reynolds number increases coefficient of lift and drag increases, while coefficient of momentum decreases.
5. The results are also represented in the form of contours for primitive variables like pressure and velocity. Velocity vectors are also plotted to know the nature of flow according in the computational domain of a aerofoil.

REFERENCES

[1] Triet MM, Viet NN, Thang PM.Aerodynamic analysis of aircraft wing.VNU J. Sci. Math. – Phys. 2015;31(2):68–75.

[2] Zhang Y, Fang X, Chen H, Fu S, DuanZ,Zhang Y. Supercritical natural laminar flowairfoiloptimization for regional aircraft wing design. Aerosp. Sci.Technol. 2015;43:152–164.

[3] Parashar H. Calculation of aerodynamiccharacteristics of NACA 2415, 23012,23015 airfoils using Computational FluidDynamics (CFD). International Journal ofScience, Engineering and TechnologyResearch. 2015;4:3.

[4] Al-kayiem HH, Kartigesh AK, Chelven K.An investigation on the aerodynamiccharacteristics of 2-D airfoil in groundcollision. Journal of Engineering Science and Technology. 2011;6(3):369-381

[5] Recktenwald, B., “Aerodynamics of a Circular Planform Aircraft”, American Institute of Aeronautics andAstronautics 022308, 2008, pp.1-7.

[6] Ravi HC, Madhukeshwara N, Kumarappa S. Numerical investigation of flow transition for NACA-4412 airfoil using computational fluid dynamics. International Journal of Innovative Research in Science, Engineering and Technology. 2013;2:7.

[7] Eleni DC, Athanasios TI, Dionissios MP. Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics

[8] Dwivedi, Y.D., Prasad, M.S., and Dwivedi, S., “Experimental Aerodynamic Static Stability Analysis ofDifferent Wing Planforms”, International Journal of Advancements in Research & Technology, Vol. 2, No. 6,June 2013, pp.60-63.

[9] Mineck, R.E., and Vijgen, P.M.H.W., “Wind-Tunnel Investigation of Aerodynamic Efficiency of Three Planar Elliptical Wings with Curvature of Quarter-Chord Line”, NASA Technical Paper 3359, October 1993, pp. 1-20.

[10] Qin, Y., Liu, P., Qu, Q., and Guo, H. "Numerical study of aerodynamic forces and flow physics of a delta wing in dynamic ground effect," Aerospace Science and Technology,Vol. 51, 2016, pp. 203-221.

[11] Nuhait, A., and Zedan, M. "Numerical simulation of unsteady flow induced by Chen, Y.-S., and Schweikhard, W. G. "Dynamic ground effects on a two-dimensional flat plate," Journal of Aircraft, Vol. 22, No. 7, 1985, pp. 638-640.

[12] Qu,Q., Jia, X., Wang, W., Liu, P., and Agarwal, R. K. "Numerical Simulation of the flowfield of an Airfoil in 1662

[13] R.Wahidi, W. Lai, J.P. Hubner, A. Lang. Volumetric three-component velocimetry and PIV measurements of Laminar Separation Bubbles on a NACA4412 Airfoil. 16th Int. Symp of Applications ofLaser Techniques of Fluid Mechanics. Lisbon, Portugal. 09-12 July, (2012).

[14] M. Agrawal, G. Saxena, Analysis of wings using airfoil NACA4412 at different angle of attack. IJMER. **3**, 1467-1469 (2013.

[15] Alex E. ockfen and Konstantin I. Matveev Aerodynamic Characteristics of the NACA 4412 Airfoil section with Flap in Extreme Ground Effect Inter J Nav ArchitOcEngng (2009) 1:1~12 11.

[16] Omar Kemal KinaciAn iterative Ocean Eng. (2014) 6:282~296.

[17] Shaowei Lia,DanjieZhoua,Yuanjing(2015) 174– 178.

[18] M. Nazmul Haquea, Mohammad Alia ,IsmatAraa Experimental investigation on the performance of NACA 4412 aerofoil with curved leading edge planform Procedia Engineering 105 (2015) 232 – 240.

[19] Abbott I, and Denhoff,A, 1959. Theory of wing section