**IOSR Journal of Engineering (IOSRJEN)** **www.iosrjen.org**
ISSN (e): 2250-3021, ISSN (p): 2278-8719
Vol. 05, Issue 1, January 2015, ||V-4|| PP 06-17

# Assessment to Domination of Precarious Air Density to the Aerodynamic Performance of Fixed Wing Aircraft

Anitha.G[1], Jagadish.R[2], Natesan.M[3*]
*[1,2,3]Department of Aeronautical, Dhanalakshmi Srinivasan College Of Engineering and Technology, Mamallapuram, Chennai, Tamilnadu, India.*

***ABSTRACT:*** *This studies have focused the unsteady aerodynamic performance of an aircraft using CFD (computational fluid dynamics). It was found that the change in density of air in flight domain influence the aerodynamic performance of aircraft in flight this result in turbulence and unsteadiness in air velocity vector. This work shows the impact of transient change in turbulence and unsteadiness in air velocity vector. This work shows the impact of transient change in air density to lift, drag and moment coefficient acting on the aircraft fuselage during flight the aim of their paper is to determine the unsteadiness in density of air influence the aerodynamic performance of aircraft with time. The compressible simulations uses performed of transonic flow with Mach number 0.84. The wing geometry is designed using the some work and CFD was developed by MATLAB code. The result is shown 0.24 arrange change of air density causes a drop in lift coefficient by an amount of 0.01014. The wing design efficiency studied is 50% and the value is low. The change in air density will cause poor aerodynamic and essentially stall.*
***KEYWORDS****: Computational fluid dynamics; unsteady aerodynamics; Aerodynamic performance; fixed wing aircraft; Angle of attack*

## I. INTRODUCTION

The aerodynamic of aircraft in flight is the interaction between the air flow and object in motion. It handles with force and moment acting on the fuselage of aircraft in relative motion with air. The aerodynamic performance like lift, drag, and moment coefficient play a vital role in evaluation of aerodynamic performance of find wing aircraft. The air is the basic flight domain for aircraft and understanding the properties of air influences. The aerodynamic behavior when the flight is critical [4]. Some examples of air properties are mass, density, viscosity and compressibility etc. Among all these properties density is most important.

The lift and drag focus are related to the mass, viscosity and compressibility of air around the airfoil surfaces. The moving aircraft are affects skin friction and wave drag by the properties of air. Skin friction will occur when the pressure change around the airfoil and this led to drag formation. The magnitude of drag force generated will depend upon the fuselage steps and degree of roughness of its surface. The smooth surface will have less drag than roughened surface.

The aerodynamic friction will occur due to pressure coefficient and drag force at the surface is referred as skin friction and is considered in calculation of drag coefficient [2]. Wave drag is a form of drag due to show wave of airfoil surfaces.

Density is defined as mass per unit volume of substances and play a role in the aerodynamic behavior of aircraft. The air is compressible due to influence of unsteady flow the density will changes. This density of air affect the skin friction and wave drag, a time dependent change of intensity of air will cause skin friction and the wave drag is change proportionally. This amount of change its corresponding effect has not been investigated from extensive search.

The main aim of this paper is to estimate coefficient will be achieved for the specified change in the density of air. In addition the lift and drag forces are operated when the body interact with the moving fluid (air) according to Newton's second law (law of momentum). The aerodynamic force changes is directly proportional with change in momentum of moving object with time by definition momentum is the product of mass and velocity [11, 12]. By converting mass of fluid to density the momentum is directly proportional to the density of fluids.

The aerodynamic performance will also depend on the wing design. This implies that unsteady air condition will influences the aerodynamic performance of an aircraft differently, based on design model. In this paper airbus 380-800 wing model was used as prototype to achieve the objectives of investigation using ANSYS fluent. Condition of this effect of unsteady air density effect will play on the relationship between changes of lift and drag coefficient with air density variation. The wing efficiency for each values of lift and drag coefficient for different angle of attack form -20deg to 50deg. When an aircraft fly pitching angles above 15deg will begin to stall.

The objectives will provide good understand of low the change in density of airflow is required to cause objectives decrease in the aerodynamic coefficient operated.

The turbulence is the main factor which cause unsteady aerodynamic behavior. Mostly commercial aircraft exposed to turbulence which will leads to the unsteady movement of fluid which cause several factor. One of the factor turbulence air flow is change in air density as time elapses. For example intertropical convergence zone (ITC2) where north and south hemisphere meets. At this point the airflow within in this region turbulence follow a thunder storm and rain fall. Aircraft flying this zone experience unsteady of aerodynamic causing the angle of attack to change with time.

The paper aims to investigate the influence of time dependent changes in the density of air to the aerodynamic performance of a fixed wing aircraft in flight.

In this paper the determination of aerodynamic performance of aircraft will based on lift, drag and moment coefficient and wing efficiency with flow of different angle of attacks. It should be noted the flow behavior of air will influenced by the dynamic change in the density of air.

The lift, drag and moment focus are shown in below

Lift $= 0.5 * C_l * p * V^2 * S$

Drag $= 0.5 * C_d * p * V^2 * S$

Moment $= 0.5 * C_m * f * d$

Where $C_L$ and $C_D$ are lift and drag coefficient dimensionless, S is airfoil surface area square unit, V is airspeed displacement /time and p is fluid density, mass/volume .

CFD involves the application to producer quantitative and qualitative predictions or analysis of fluid flow. The analysis of CFD is based on conservation laws such as conservation of momentum and momentum and energy.

CFD simulation apply two dominant approaches that is finite difference and finite element simulation which will consider later. CFD is discretization of spatial domain into mesh point. The flow condition adopted in this paper was compressible fluid flow for that density of air will change with time for high $R_e$ (Reynolds number) and transonic flow (M = 0.94).

## II. METHODOLOGY

**CFD simulation**

CFD was used as the tool in ANSYS fluent version. This paper concludes the lift and drag of the model design at different angle of attack to obtain stall angle of attack for wing design and transient simulation so aimed at analyzing the influence of unsteady air density of lift and drag coefficient acting on surface of airfoil.

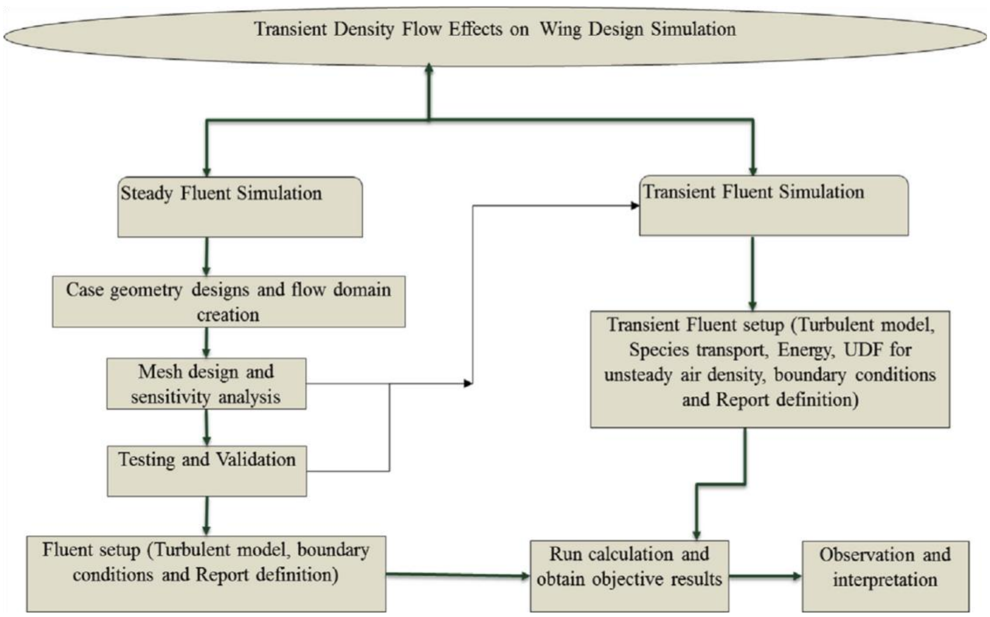



**Fig.1 Flow chart showing step-wise procedures followed to obtain objective results.**

**Geometry creation**

The fig 2 show the top, side and front view of the model wing designed by SOLIDWORK software. The geometry of wing designs of symmetry airfoil of fuselage. Structure similar to Airbus 300-800 model.

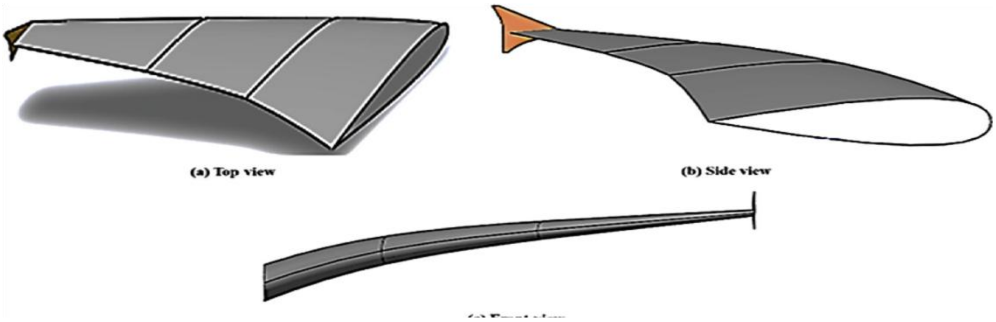



**Fig .2. Top view and front view of the wing design-1 model geometry**

**Flow domain certain**

To study the fluid flow behavior across the airfoil surface a flow in a cuboid shape was created using ANSYS Geomedeler. The wing model was enclosed within the body of influence before it was attached to the domain geometry. The body influence was introduced to control the mesh size and proximity at the wing geometry sensitive to flow. It is easier to control the mesh sizes for geometry with multiple intersecting closed bodies than using boundary layer. The domain fig 3.

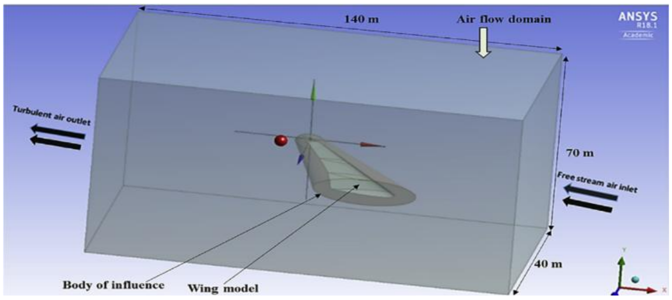



**Fig 3. Cuboid-shaped air flow domain enclosing the wing geometry.**

It is assumed the geometry shape flow domain unit have no domain the aerodynamic coefficient at the wing surface. Notwithstanding the fact that it is important to ensure that the size of domain should be large to prevent the reverse flow at high pressure all zone and wing tip vortices.

At the wing tip the fluid recirculates by doing this create the increases of drag. Mostly this problem is neglected. But may introduce error in the solution. The flow around two wing tip do not interference because this includes in difference direction. When flow attain low pressure zone a reversed flow is observed.

The swirl dissipation rate is slow and will linger in the flight domain even after the aircraft has moved on. Hence produce turbulence which may be determined to nearly aircraft flying in same pitch.

The body of influence is an advanced explicit sizing control using ANSYS global mesh control system to resolves any accurately gradient flow area (at airfoil surface) such as flow separation, recirculation, reattachment or wake zone. This provide a reliable prediction of flow streamlines, aerodynamic forces and moment at airfoil surface.

For a viscous flow no slip wall boundary is recommended in the boundary condition. It is important because it assume the wall boundary is identical to the velocity of this wall. If this axis zero is no relative movement between models geometry wall boundary and the fluid.

The choices of domain size is important and significant impact on the flow fractures prediction especially on the wall boundary. The domain size should be large to eliminate wall effect. To eliminate the wall boundary effect a slip wall boundary condition with zero shear option using symmetry boundary is a suitable. Alternately no slip wall boundary layer can be used on recommendation by Rapp.

This size of the flow domain considered in this study was made moderate with dimension.

In this simulation process the number of iteration used for each run equal 5000, to ensure residuals converged. This is for transient simulation 2000 iteration for the steady state simulation and 10 hour computational times was used for every run. This is due to low computing power of computer used to carry out this study. To avoid wall boundary effect domain size and slip boundary symmetrical was will no shear was used.

**Mesh Creation and Sensitivity Analysis**

Model meshing or mesh creation is the next preprocessing step after the model geometry has been created. The discretization of the fluid flow domain is referred as mesh or grid generation. There exist 2 Category of mesh generation normally structured and unstructured mesh. The type of mesh are generated based on the topology of the model geometry the main difference between the structured and unstructured mesh is use the hexahedral element with implicit connectivity of point in the mesh in the mesh and tetrahedrons element with an explicit defined mesh connectivity respectively.

**The final mesh design**

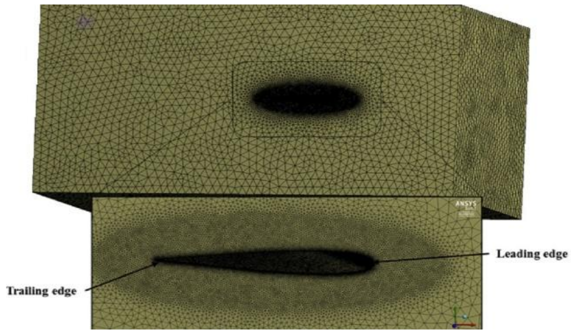



**Fig. 5. Fig showing studied viewing of the dense mesh created at airfoil edge and around the surface area**

It is an unstructured mesh created with ANSYS meshing using patch/body conforming. So no grid surface with each other at boundary surface. In addition the unstructured surface and volume meshing was created using the Delaunay algorithm to connect point and recover boundary edges. As such the domain is at composed into interactives subdivisions, local clustering and triangular mesh can be used further analysis.

A key advantage of unstructured meshing over structured is its ability to allow point insertion response to high flow gradient but difficulties can be experienced in resolving strong flow fractures such as vortices and wake requiring longer flow solution run time.

Mesh sensitivity is an important part of evaluating the mesh quality in CFD simulation. This involves mesh size and running the simulation until critical objectives result.

The fig 4 grid sensitivity plot of describing how the number of element whiles increased up accuracy of the CFD result as the sizes of element whiles increased up to 15 million element , the lift , drag , and moment coefficient values converged to the 4$^{th}$ order accuracy (approximately 0.00043 mean error).

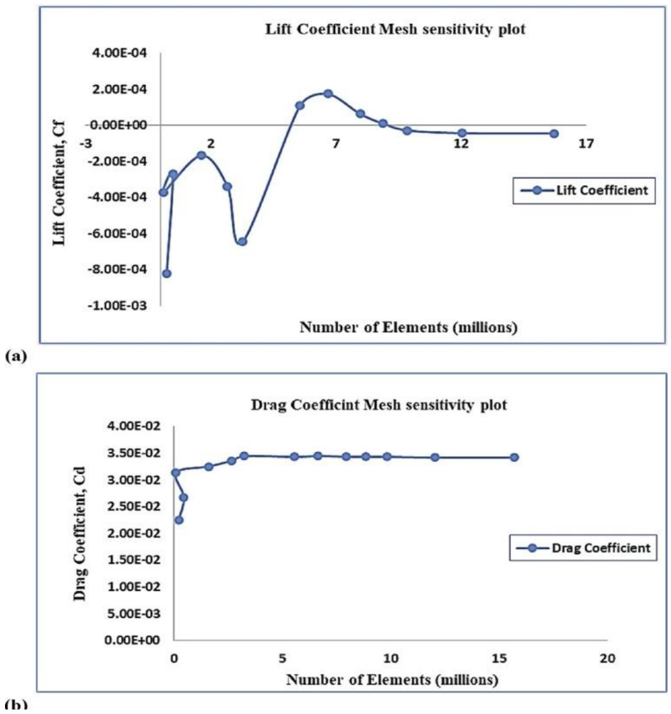



**Fig. 4. (a): mesh sensitivity plot for lift coefficients at 0angle of attack. (b): mesh sensitivity plot for drag coefficients at 0angle of attack.**

**Table 1 Wing Design Parameters**

| Design Considerations | Data |
|---|---|
| Mean Camber Line | 2.1 m |
| Maximum Camber Thickness | 0.22 m |
| Chord Length | 2.1 m |
| Wing Span Area | 95.2 m square |

**Table 1 provide important parameter used in design of wing model**

**Table 2: Final mesh design settings.**

| Parameters | Type |
|---|---|
| Mesh Type | Unstructured Mesh |
| Method or Element Type | Tetrahedrons |
| Physics Preference | CFD |
| Algorithm | Patch Conforming |
| Number of edge Sizes | 1000 |
| Local Minimum Size | 1.4e-002 m |
| Size Function | Proximity and Curvature |
| Face Meshing Element Size | 0.1 m |
| Mesh Quality or Skewness | 0.86 (<0.9) |
| Calculated $y^b$ Value<br>Total number of Nodes | 0.99851<br>2,780,716 |
| Mesh Curvature Angle | 2 |

Table 2 show the final mesh design parameter such as edge, face sizing,Size function, local mesh size and calculated $g^+$ value around the airfoil.

**Pre-Processor Settings**

Turbulent model testing and selection criteria. The selection of the turbulent model is another important for CFD simulations because flow turbulent models are selected based on specific computational objectives. The model common turbulent model used on CFD simulation are based on Reynolds average naiver stroke equation.

In this paper different turbulent model were investigated for convergence and variation of objectives result with data published. This was aimed at investigating how each turbulent model converged to the expected result so as to select a turbulent model that will yield accurate result. This verification is published using the NACA B737 airfoil root data. The result obtained from this analysis show k-ε RNG convergedbetter compared to other turbulent model. This can see in fig 6.

Critical analysis of the plot in fig 6(a) show the drag coefficient prediction obtained with standard K-ω turbulent model correlated better with experimental data.

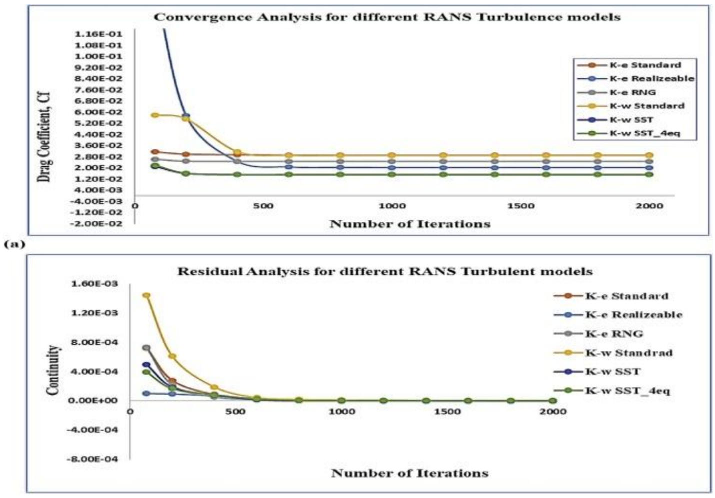



**Fig. 6. (a): convergence testing for RANS turbulent models. (b): Residual analysis for different RANS turbulent models**

The hydraulic diameter and turbulent intensity were calculated from equation 1 & 2

$D_H = 2ab/ (a+b)$
$I = 0.16\ Re_N^{-1/8}$
$D_H$ is hydraulic diameter inmeter.
I is the turbulent intensity in %.( percentage)
Re in Reynolds number dimensionless

**Numeric and Discretization Scheme**

ANSYS software make use of two types of solvers that is pressure based and density based solvers. In past, pressure based solvent is used for low-speed incompressible flow and assume change in density is negligible and density based solved is for high-speed compressible flow advance in ANSYS software version 18.1 has extended the applicability of these solvents such as pressure-based solver which has low computational cost, time and power can be applied to solve momentum and pressure equation for compressible flow by de-coupling both equation.

Also pressure-based segregated algorithm solvers compute pressure correction equation to correct mass fluxes, pressure & velocity field and solver. Scalar equation such as turbulent, species, transport and energy.

The above statement, the standard k-w turbulent model is known to predict accurately pressure gradient and skin friction coefficient along wall boundary and not require wall function as realizable k-w turbulent model.

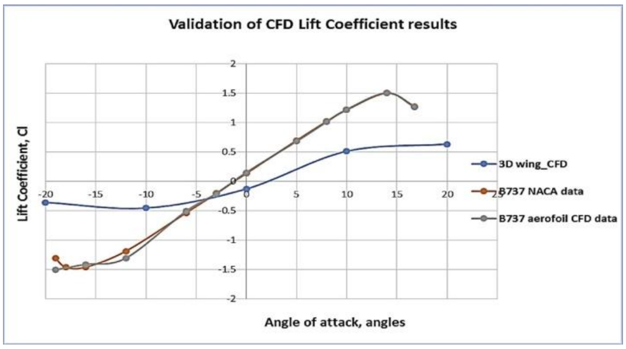
Fig (6b) is a plot showing how the residual are converging with increase in number of iterations. In this setting the convergence criteria where reduced to fourteen significant figure to allow the residual to converge with aim of minimizing round-off.
The spatial discretization scheme, the pressure, momentum & energy equation calculation was set to second-order to increase the accuracy of CFD prediction. The pressure velocity coupling in the solution method was set to SIMPLE to decouple & solve both equation independently.

**Material and Boundary Condition**

For the CFD simulation the Table 3 show the fluid data and boundary condition parameters. It is noted that a steady and transient simulation is conducted to observe the fluid pattern around the wing geometry. In this material section uses defined function developed explicitly using MATLAB programcode was run in fluent. Environment with to enable changes in air density with flow time during the transient simulation
Fig. 7. Validation plot of lift coefficient versus angle of attack.

**Qualitative CFD Result Analysis**

It is a fundamental process to ensure reliable result. This is key part of CFD simulation quality control. Verification is a process to ensure that the implemented model and solution accurately represent actual develops concepts & include mesh sensitivity, solves and turbulent models applicability and validation involves calculating CFD result with experimental data describing the applicability of C*D to solve real-world problems. A 2D & 3D simulation was conducted & validated in B737 NACA X-foil data to ensure choice of turbulent model will produce reliable result. It demonstrate a good calculation of CFD result with B737 X-foil data.



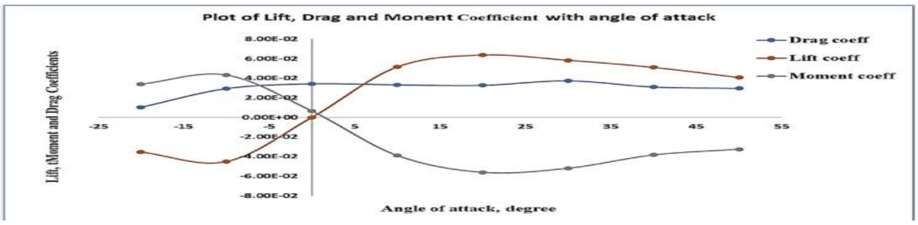




**Fig. 8. Plot showing the relationship between lift, drag and moment coefficient at varying angle of attack. Validation of 2D & 3D CFD Simulation**

In this paper boundary condition were changed to suit that simulation case to be used for validation purpose. Here NACA airfoil data were used as base case. In plot in fig 7 show simulation result correlate well with NACA aero foil experimental data implying the simulation setup was properly done.

The plot shows that how the density of air within the flow domain changes with time. It said that for every 125 flow period. The density of air will change by an account of 0.2 kg/cm. the studying shown that change in a property of air can influence the speed of moving object.Form the plot the flow pattern in non-linear or parabolic behavior is observed.

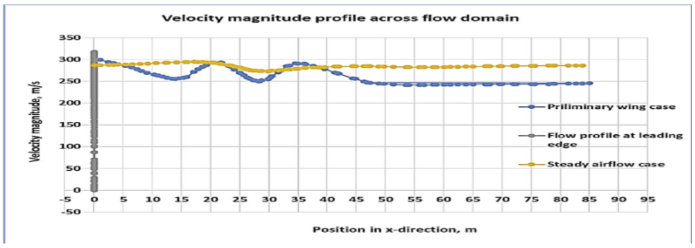



**Fig. 10. Velocity magnitude flow profile for Unsteady and steady airflow.**

Fig 10 demonstrate how density of air changes with time. Re –arranging equation 1 & 2 the square of the velocity of the body will vary inversely proportional to the density of fluid. In this analysis the density of air will cause an inverse proportional change to the velocity of aircraft motion. It is based on the theory when density change the turbulence intensity will also change the unsteady flow condition will influence the aerodynamic stability of an aircraft.

In this paper the total flow time of 200s was consider. This is hypothetical value used in this investigation to enable successful achievement of objectives results.

**Analysis of Aerodynamic Coefficient with Angle Of Attack**

To perform steady state flow simulation before considering the effect of time to the aerodynamic coefficient obtained. Fig 8 show that the how to lift, drag and moment coefficient change with angle of attack range between 20–50.

It is observed that maximum lift was obtained at approximately 18 angle of attack similar to result obtained by neccietal (2009), hull (2007) and liu et al. (2015). When lift generate to decrease after 20deg angle of attack where the aircraft stall due to airfoil separation.

The moment coefficient plot is linear with negative gradient result range between -10deg to 20deg angle of airfoil separation. This signifies that aircraft will experience stable flight during steady flow condition. The moment coefficient will be positive during unstable flight from the fig 12 show section analysis of aerodynamic coefficient with flow and time below.

**Relationship between Flow Time and Density of Air**
Recall the UDF was introduced into ANSYS fluent environment to cause density of air within the flow domain.

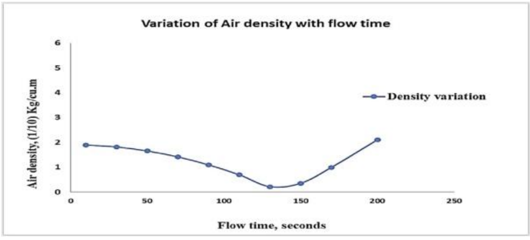



**Fig 9: variation of air density with flow time**

**Analysis of Velocity Magnitude with Position**
Fig 10 show how the velocity magnitude of aircraft changes with position. Two cases were considered in this analysis which are investigating the effect of steady and unsteady flow condition on the aircraft velocity magnitude.
From a critical analysis of fig.10. It is observed that a significant change in aircraft velocity was experience for the unsteady state result compared with steady state.
The estimated change in the velocity magnitude was about 50m/s on average for the unsteady simulation case. Further change in the speed of the aircraft can reduced the lift and increase the drag this has investigated in (fig 12 a & b).
Comparing fig (12 c & a) it is observed that the lift coefficient is inversely related to moment coefficient is inversely related to momentcoefficient. Taken into consideration result for 18deg angle of attack. The aircraft achieved a lift obtained and the moment coefficient equaled 0.01. This is due to aircraft pitches upward. The moment coefficient is negative and vice versa.
Adding to the velocity profile result above fig 11 demonstrate below shock waves are produced of the point where travelling free airstream comes in contact with leading edge. At the time airflowseparation will experience the leading edge curvature will cause airspeed to be higher at the upper surface that lower static pressure is generated at the contact point will continue for long way to contribute to drag.

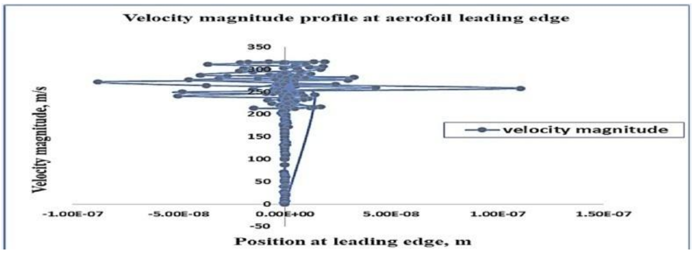
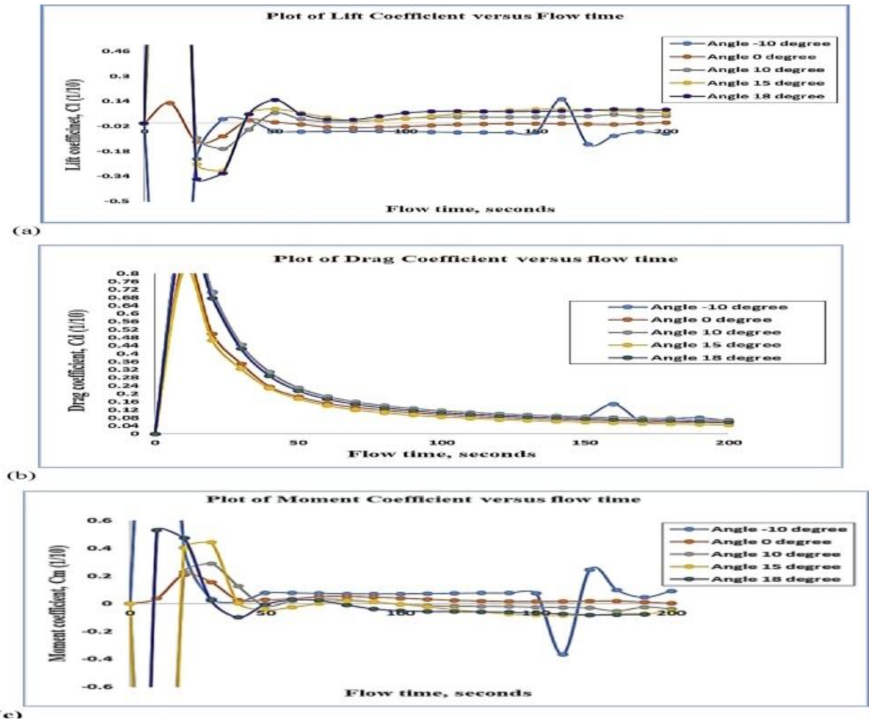



**Fig. 11. Plot showing shock waves produces at the airfoil leading edge**
**Analysis of Aerodynamic Coefficient with Flow**

**Time**
The aerodynamic performance analysis is the function of lift, drag and moment forces acting on the wing of an aircraft in flight. Calculation of these force depend on airspeed parameter coefficient and density according to equation.




The plot fog 12 a – c demonstrate the lift, drag and moment coefficient generated at the wing changes with time and angle of attack note density of air changes with time as observed in (fig 9 above). The indicated the purpose of this study and implies that the change in the density of air to impact on the lift, drag and moment generated.

From literature when the lift coefficient increase with increase of angle of attack. The drag coefficient is expected to decrease relatively. Fig 12 shows the drag coefficient decreases with increase in flow time the slope of this curve is steep within flow period 10 and 50s because this represent the initial stage when the aircraft begins to experience unsteady airflow condition and later stabilizer after 200s.

A change in the density of air by an amount of 0.24 kg/m$^3$ caused the lift coefficient to reduce by an amount of about 0.23, 0.014, 0.0220, and 0.048 of angle of attack 10, 0, 10, 15, and 18respectively at the end of 200 s flow time. This was calculated the difference between maximum and minimum value obtained from plot.
The moment coefficient is important parameter determine whether the nose of an aircraft is pitching upward or downward when movement coefficient is negative the aircraft is observed as pitch upward and vice versa. Considering the result shown in fig C the moment coefficient does not converge set for all angle of attack. This significance the aircraft is unstable motion with time due to influence of unsteady airflow condition. This behavior was determined to the structural integrity of an aircraft and passenger safety during flight.

**Analysis of Wing Efficiency**

The wing efficiency of an aircraft is ratio of lift coefficient and drag coefficient and the basis can determine the aerodynamic performance of an aircraft.

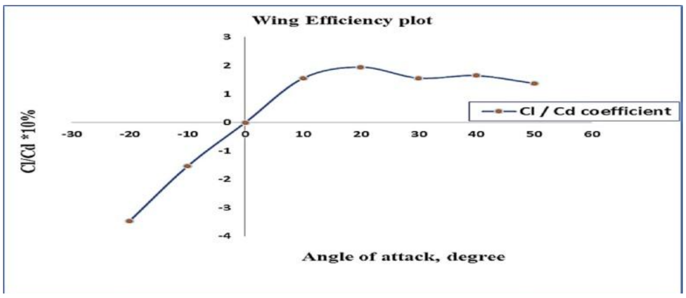



**Fig. 13. Variation of wing efficiency with the angle of attack.**

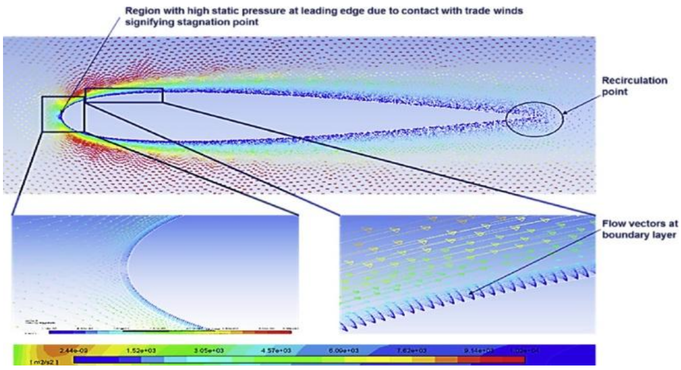



**Fig. 14. Flow vectors around the airfoil wall boundaries in 2-D.**

Fig 13 above show the wing efficiency change with different angle of attack. The main aim of wing is to generate lift and high wing efficiency above 50 % is necessary to improve the unsteady aerodynamic performance of an aircraft within unfavorable airflow condition when the efficiency of wing low (<50%). The aircraft will experience poor aerodynamic stability in unsteady airflow condition.

From this study it can deduced that the maximum wing efficiency achieved from 56% at 18deg angle of attack. According to necci et al (2009) the aircraft will attain maximum lift at an angle within range 15deg to 20deg depend on wing design A 56% wing efficiency within this range is poor and imply the aircraft was to have poor aerodynamic performance. This statement is proven from result obtained fig 8 and 12(a) above.

**Qualitative CFD Result Analysis**

Flow velocity magnitude contours. From observation of fig 14 above show airflow velocity vector at the wall boundary around airfoil in 2D. The vector density at the wall is denser than outside the boundary region. This is as a result of the boundary layer creates around the airfoil boundaries.

The airfoil boundary represent a sensitive flow gradient region and is important to ensure that the boundary layer thickness law such that $y^+$ value less than 1. When using k-w turbulent model. The $y^+$ obtained at boundary layer in a mesh design was within the recommended limit (see table 2.1)

Visualizing flow pattern in fig 15. It is evident the airflow path is steady or linear from fig 15(left) parabolic from fig 15(right) caused by the UDF implemented. The steady flow vector was desired from steady-state CFD simulation where density of air is turbulent (incompressible flow) and compressible for dependent CFD simulation. In addition airfoil is observed to be have better aerodynamic performance during steady state condition than the flow was unsteady. This is as result of the affective airflow velocity profile shown in fig (15a).

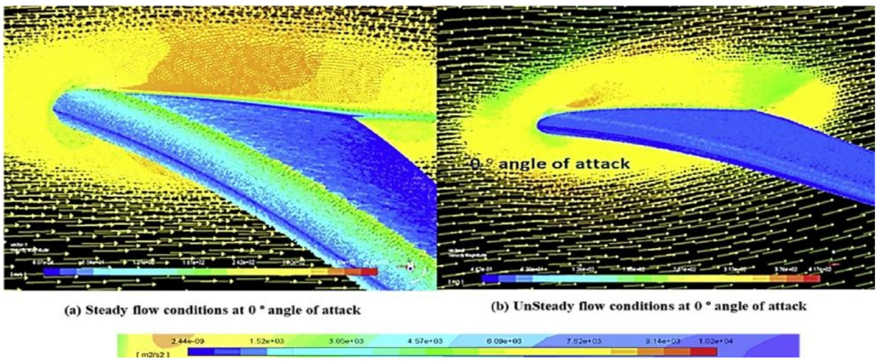



**Fig 15. Flow velocity magnitude vectors around the airfoil in 3D.**

## III. CONCLUSION

The following conclusion were made from this study. That is

- The change in density of air by amount of 0.24 kg/m$^3$ caused the lift coefficient is reduce by an amount of 0.23, 0.014 0.022, 0.042, and 0.048 at angle of attack -10deg, 0deg, 15deg, and 18deg respectively at the 200s flow time.
- The drag coefficient is decrease with increase of time. The unsteady effect of air density caused because of the aircraft to tendency of pitch upwards and downwards for 200s flow time.
- The angle of attack and maximum of 56% efficiency was obtained at 18deg angle of attack.

**REFERENCES**


[1]. F. Alobaid, B. Epple, Improvement, validation and application of CFD/DEM model to dense gas¯solidflow inafluidized bed, Particuology 11 (5) (2013) 514¯526.

[2]. J. Ballmann, A. Dafnis, H. Korsch, C. Buxel, H.G. Reimerdes, K.H. Brakhage,

[3]. H. Olivier, C. Braun, A. Baars, A. Boucke, Experimental Analysis of High Reynolds Number Aero-Structural Dynamics in ETW, AIAA, Reston, VA, USA, 2008. AIAA Paper 2008-0841.

[4]. M.H. Dickinson, K.G. Gotz, Unsteady aerodynamic performance of model wings at low Reynolds numbers, J. Exp. Biol. 174 (1) (1993) 45¯64.

[5]. Engineering ToolBox, Air - Altitude, Density and Specific Volume, 2003 [Online] Available at: https://www.engineeringtoolbox.com/air-altitude-density-volume-d _195.html. (Accessed 17 July 2018).

[6]. O.S. Gabor, A. Koreanschi, R.M. Botez, Low-speed aerodynamic characteristics improvement of ATR 42 airfoil using a morphing wing approach, in: IECON 201238th Annual Conference on IEEE Industrial Electronics Society, IEEE,2012, pp. 5451¯5456.

[7]. O.S¸. Gabor, A. Koreanschi, R.M. Botez, M. Mamou, Y. Mebarki, Numerical simulation and wind tunnel tests investigation and validation of a morphing wingtip demonstrator aerodynamic performance, Aero. Sci. Technol. 53 (2016) 136¯153.

[8]. K. Gharali, D.A. Johnson, Dynamic stall simulation of a pitching airfoil under unsteady freestream velocity, J. Fluids Struct. 42 (2013) 228¯244.

[9]. Z. Guo, T. Hirano, R.G. Kirk, Application of CFD analysis for rotating machinery: Part1¯¯hydrodynamic, hydrostatic bearings and squeezefilm damper, in: ASME

[10]. Turbo Expo 2003, Collocated with the 2003 International Joint Power Generation Conference, American Society of Mechanical Engineers, 2003, pp. 651¯659.

[11]. M. Ghoreyshi, I. Greisz, A. Jirasek, M. Satchell, Simulation and modeling of rigid aircraft aerodynamic responses to arbitrary gust distributions, Aerospace 5 (2) (2018) 43.

[12]. S. Ho, H. Nassef, N. Pornsinsirirak, Y.C. Tai, C.M. Ho, Unsteady aerodynamics and flow control forflapping wingflyers, Prog. Aerosp. Sci. 39 (8) (2003) 635¯681.

[13]. A. Jameson, M. Fatica, Using Computational Fluid Dynamics for Aerodynamics–A Critical Assessment, Stanford University, California, United States of America, 2017, pp.5¯8.

[14]. A. Jameson, M. Fatica, Using Computational Fluid Dynamics for Aerodynamics, 2006, pp.1¯4.

[15]. A. Stefan, C. Larco, R.C. Pahonie, I. Nicolaescu, Coupled transient analysis of a UAV composite wing, Scientific Research&Education in the Air Force-AFASES 2 (2015).

[16]. Z. Liu, W. Liu, X. Nie, G. Yang, Computational-fluid-dynamics solver with preconditioned method for aerodynamic simulation of high-lift configuration, J. Aircr. 53 (4) (2015) 1056¯1064.

[17]. Z.J. Wang, J.M. Birch, M.H. Dickinson, Unsteady forces andflows in low Reynolds number hoveringflight: two-dimensional computations vs robotic wing experiments, J. Exp. Biol. 207 (3) (2004) 449¯460.

[18]. C. Necci, N. Ceresola, G. Guglieri, F. Quagliotti, Industrial computationalfluid dynamics tools for the evaluation of aerodynamic coefficients, J. Aircr. 46 (6) 1973¯1983.

[19]. Z. Liu, W. Liu, X. Nie, G. Yang, Computational-fluid-Dynamics solver with preconditioned method for aerodynamic simulation of high-lift configuration, J. Aircr. 53 (4) (2015) 1056¯1064.

[20]. A. Koreanschi, O. Sugar-Gabor, R.M. Botez, Numerical and experimental validation of a morphed wing geometry using price-païdoussis wind-tunnel testing, Aeronaut. J. 120 (1227) (2016) 757¯795.

[21]. C. Necci, N. Ceresola, G. Guglieri, F. Quagliotti, Industrial computationalfluid dynamics tools for the evaluation of aerodynamic coefficients, J. Aircr. 46 (6) 1973¯1983.

[22]. A. Jameson, June. Time dependent calculations using multigrid, with applications to unsteadyflows past airfoils and wings, in: 10th Computational Fluid Dynamics Conference, 1991, p. 1596.

[23]. I. Mary, P. Sagaut, Large eddy simulation offlow around an airfoil near stall, AIAA J. 40 (6) (2002) 1139¯1145.

[24]. C. Rumsey, J.P. Slotnick, M. Long, R.A. Stuever, T.R. Wayman, Summary of thefirst AIAA CFD high-lift prediction workshop, J. Aircr. 48 (6) (2011) 2068¯2079.

[25]. M. Sun, J. Tang, Unsteady aerodynamic force generation by a model fruitfly wing inflapping motion, J. Exp. Biol. 205 (1) (2002) 55¯70.

[26]. D.G. Hull, Fundamentals of Airplane Flight Mechanics, Springer, Berlin, 2007, p. 15.

[27]. B.E. Rapp, Microfluidics: Modeling, Mechanics and Mathematics, William Andrew, 2016. Available at: https://www.sciencedirect.com/topics/engineering/no-slipboundary-condition.

[28]. J.L. Reel, N.D. Baltadjiev, Using computationalfluid dynamics to generate complex aerodynamic database for VTOL aircraft, in: 2018 Applied Aerodynamics Conference, 2018, p. 4211.

[29]. G. Rozza, H. Malik, N. Demo, M. Tezzele, M. Girfoglio, G. Stabile, A. Mola, Advances in Reduced Order Methods for Parametric Industrial Problems in Computational Fluid Dynamics, 2018 arXiv preprint arXiv:1811.08319.

[30]. B.N. Hanna, N.T. Dinh, I.A. Bolotnov, High-Fidelity Simulation-Driven Model Development for Coarse-Grained Computational Fluid Dynamics (No. INL/CON-1637736), Idaho National Lab.(INL), Idaho Falls, ID (United States), 2016.