Design and Analysis of Non Planar Wing in Commercial Aircraft

Gobpinaath M
UG student, Department of Aeronautical Engineering, Hindusthan college of Engineering and Technology,Coimbatore-641032
Sivajiraja K
UG student, Department of Aeronautical Engineering, Hindusthan college of Engineering and Technology,Coimbatore-641032

Suresh C
Assistant Professor, Department of Aeronautical Engineering, Hindusthan college of Engineering and Technology,Coimbatore-641032

Ramesh K
Assistant Professor (Sr.G), Department of Mechanical Engineering, Government college of technology, Coimbatore 641032

Abstract :- From the development of the first powered flight (1903) to the present time, the study of the aerodynamic design has played an important role in the airplanes optimization. In the current era of globalization every prosperous nation in this world wishes to develop a fast moving aircraft with a high lift to drag co-efficient. Non-planar wing configurations promise a significant improvement of aerodynamic efficiency and are therefore currently investigated for future aircraft configurations. The purpose of this project is to maximize the lift for a given amount of drag and perform force (lift and drag) measurements on a non planar C-Wing with a supercritical airfoil NACA 64-21 (Airbus A300) and compare the results obtained with those corresponding to a conventional planar Airbus A300 wing.

Two approaches using Computational Fluid Dynamics and wind tunnel tests have been followed. Three dimensional computational fluid dynamics (CFD) analyses of four different C-Wing geometries with mounting angle (90º,95º,100º,105º)with the same aspect ratio as that of a planar wing is carried out. The second approach is to incorporate a C-Wing with a mounting angle of 90º between the horizontal and vertical winglet into a prototype and measurements are taken by wind tunnels. Comparison of the aerodynamic forces obtained from the preliminary wind tunnel test results and CFD analysis are showing good agreement between them and strongly suggest that the C-Wing do possess capabilities to reduce the induced drag significantly, which could be owed to the reduction in the downwash of the complete configuration and hence improve the C_L/C_D ratio.

Keywords: C-Wing, Non-planar wings, Angle of Attack, CL/CD, Induced Drag, Mounting angle.

I. INTRODUCTION

Aircraft performance is highly affected by induced drag caused by wingtip vortices. Winglets, referred to as vertical or angled extensions at aircraft wingtips, are used to minimise vortices formation to improve fuel efficiency. Winglets application is one of the most noticeable fuel economic technologies on aircraft. Winglets, defined as small fins or vertical extensions at the wingtips, improve aircraft efficiency by reducing the induced drag caused by wingtip vortices, improving the lift-to-drag ratio (L/D). Winglets function by increasing the effective aspect ratio of the wing without contributing significantly towards the structural loads. The winglet concept was first developed in the late 1800's by Frederick W. Lanchester. His investigations proved that under high lift conditions, wingtip drag can be reduced by placing a vertical surface at the wingtip.[15]

Conventional winglets provide maximum drag cutback and improve L/D under cruise conditions only. During non-cruise conditions, these winglets are less likely to improve aircraft performance and subsequently, they do not provide optimal fuel efficiency during take-off, landing and climb. Non-cruise flight conditions add up to a significantly large fraction of a flight and therefore, winglet designs must be optimized to be able to function during both cruise and non-cruise flight conditions. Research on conventional winglets improvement methods have been more dominant compared to any other types of winglet. In recent years, extensive research has been increased to boost the aircraft performance during flight.

A. Non-Planar wing:

Non-planar wing planform is a closed wing that uses a continuous surface, eliminating the wing tip. Non-planar wing can be thought of as the maximum expression of a wing tip device, which has the aim of eliminating the influence of the wingtip vortices which occur at the tips of conventional wings. Non Planar wings have an advantage of reducing the induced drag without compromising with high aspect ratio keeping the aircraft small in size, suitable for a reconnaissance UAV like in the AAI Aerosonde. It has been attempted to replace the mono wing design with non-planar wings Dayton, Kroo, etc. This involves multi layered wings or a single wing which is not in a single plane. Nonplanar wings include configurations such as biplanes, box-planes, ring-wings, joined wings, and wings with winglets.Non-planar wings are generally used for reducing the induced drag. According to Kroo the vortex drag of a commercial airplane constitutes to as much as 40% of the entire drag during cruise and as much as 80% - 90% of the total drag during low speed conditions like climb and take off.[10] Nonplanar wings offer the possibility of reduced drag compared with planar wings of the same span and lift. Apart from configuration differences related to stability and trim, variations in nonplanar geometry represent one of the few major differences in aircraft conceptual design. Such designs may be of interest because of their potential for lower vortex drag at a fixed span, a key constraint for many aircraft, including very large commercial transport concepts. However, several non-aerodynamic features are of interest as well including effects on stability and control, characteristics of wake vortices, and structural implications of the nonplanar design.

This project reviews some of the concepts that have been pursued and discusses some of their possible advantages or disadvantages. We consider the potential of some of the concepts to improve performance incrementally or to change the configuration significantly.

Nonplanar wing concepts may be divided into a few categories based on their primary geometric or aerodynamic characteristics. These include:

- Multiplanes (biplanes, triplanes)
- Closed Systems (box planes, ring wings)
- Strut-Braced Systems (Lifting struts, joined wings)
- Nonplanar monoplanes (wings with winglets and other tip devices)
- Planar wings with nonplanar wakes (Crescent wings, Split-tips)[1]

Multiplanes:

Fig.1 (a) shows the multiplane configuration. The induced drag of a multiplane may be lower than that of a monoplane of equal span and total lift because the nonplanar system can influence a larger mass of air, imparting to this air mass a lower average velocity change, and therefore less energy and drag. For a biplane, if the two wings are separated vertically by a very large distance, each wing carries half of the total lift, so the induced drag of each wing is 1/4 that of the single wing. The inviscid drag of the system is then half that of the monoplane.[1]

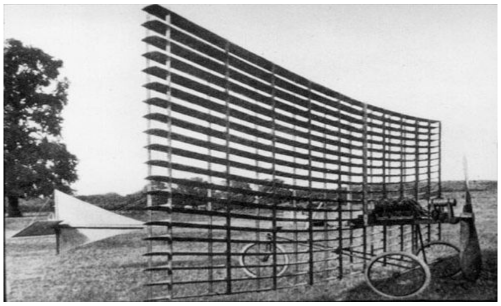


Fig1 (a) Multiplanes

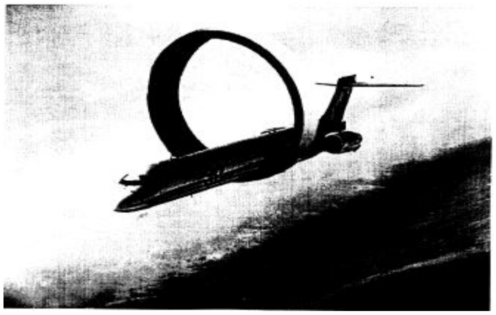


Fig.1 (b). Closed wing

CLOSED SYSTEMS:

Fig.1 (b) shows the closed wing configuration. The aerodynamics of nonplanar wing systems that form closed loops are very interesting. Such configurations include box-planes, ring wings, joined wings, and "spiroid-tip" devices. Wings that form closed loops, such as the ring-wing illustrated below, do not eliminate the "tip vortices" or trailing vortex wakes even though the wing has no tips. Still, the vortex drag of the circular ring wing is just 50% that of a planar wing with the same span and total lift and the concept has been studied at several organizations, including early aviation pioneers, a major aircraft manufacturer, as well as several toy companies.[1]

Strut-braced system:

Fig.1(c) shows the strut-braced system. Aircraft concepts that employ auxiliary aerodynamic surfaces as struts to improve both aerodynamic and structural efficiency have been studied extensively.[16]

- In joined-wing designs (below) the horizontal tail sweeps forward and joins the main wing, forming a strut. The tail is then in compression, reducing wing bending moments. If the tail is large enough to be positively loaded, some induced drag savings is achieved, while if it is carrying a down-load, the closed loop feature of the system minimizes trim drag. The concept was studied by Boeing as a radar platform and by others as a commercial transport.
- Pfenninger's laminar designs with lifting struts exploit the nonplanar strut geometry primarily for structural weight and stiffness, although some induced drag reduction may be achieved.[1]

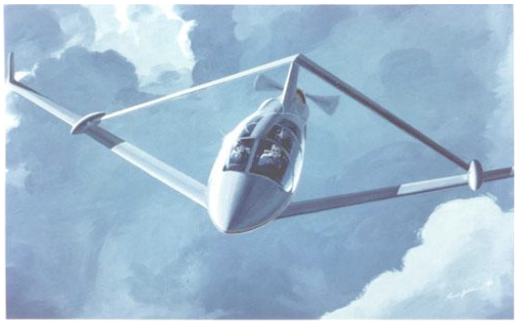
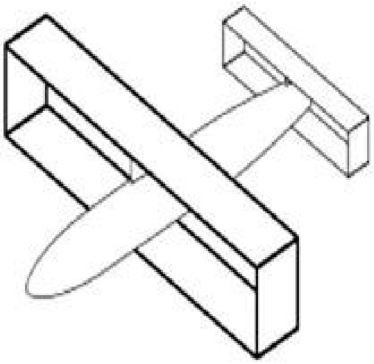


Fig 1 (c)Strut- braced system



(d) Box wing

Box wing:

A Box wing configuration shown in fig.1(d) assures saving in fuel consumption because of lower induced drag. This reduction in induced drag is a result of higher glide ratio when compared to conventional monoplane design. The box wing configuration has proven to consume 9% less fuel as a result of 14% increase in the glide ratio. Also, because of the improved lift-to-drag ratio, we can expect lesser noise by the UAV making the surveillance mission without making noise.[10]

B. Why Non-planar Wings used ?

The vortex drag of commercial aircraft accounts for a large fraction of airplane cruise drag (typically about 40%) and therefore concepts that result in reduction of vortex drag may have a significant effect on fuel consumption, the hundreds of millions of dollars spent annually by airlines on fuel, and its effect on the environment. Vortex drag is even more significant at low speeds where vortex drag typically accounts for 80%-90% of the aircraft’s climb drag at critical take-off conditions. Although one might argue that take- off constitutes a very small portion of the flight, its influence on the overall aircraft design is profound. Since conditions associated with engine-out climb shortly after take- off are often critical constraints in the aircraft design, changes in aircraft performance at these conditions influence the overall design and so have an indirect, but powerful, effect on the aircraft cruise performance. While a 1% reduction in drag due to lift might improve the cruise lift-to-drag ratio by 0.4% with a similar effect on range, the improved low speed climb performance may make it possible to achieve acceptable take-off and climb with almost 1% greater take-off weight, leading to an increase in range several times that associated with the simple cruise L/D improvement. As a result, drag due to lift has a much greater significance to aircraft performance than might be inferred simply from the aircraft cruise aerodynamics. Furthermore, even for aircraft that are not constrained by a required climb gradient, lower drag at high lift conditions leads to reduced noise.

Of course, induced drag may be easily reduced by increasing the span of a planar wing. A 10% increase in wing span leads to a 17% reduction in vortex drag at fixed speed and lift. A primary reason that wing spans are not increased to reduce drag is that the higher structural weight and cost make such efforts counterproductive. Nonplanar wing concepts must therefore be assessed similarly, taking into account more than just the potential improvements in cruise aerodynamics.

In fact, some unconventional nonplanar aircraft concepts are promising more because of their structural characteristics than their aerodynamic features. Some designs exploit the nonplanar geometry to improve effective structural depth of the wing system and can achieve drag reduction indirectly by using the improved structural efficiency to accommodate larger spans without higher structural weight. Other design concepts utilize the differences between non-planar and planar wing load variation with lift coefficient to reduce structural loads at critical conditions and again save in weight or add span at fixed weight.[3]

- In coefficient form , drag can be written as

- $C_D = C_{DO} + C_{Di}$
- C_{DO} – Zero lift drag C_{Di} – Induced drag

➢ The coefficient of Induced drag component can be written as

- $C_{Di} = C_L^2 / \pi.e.AR$
- e – Oswald efficiency factor or span efficiency

Induced drag can be minimized by

- ➢ Decreasing the coefficient of lift
- ➢ Increasing the aspect ratio
- ➢ Increasing the span efficiency

Although increasing span has aerodynamic benefits , structurally it is not a best option, as in increase result in aero elastic problems on the wing . It also increases the airport terminal restriction.

Another approach for induced drag reduction can be obtained by increasing the span efficiency factor . The planar wing having a given span and lift, the maximum span efficiency factor is limited to unity . For non planar systems the span efficiency factor can be increased beyond unity.[2]

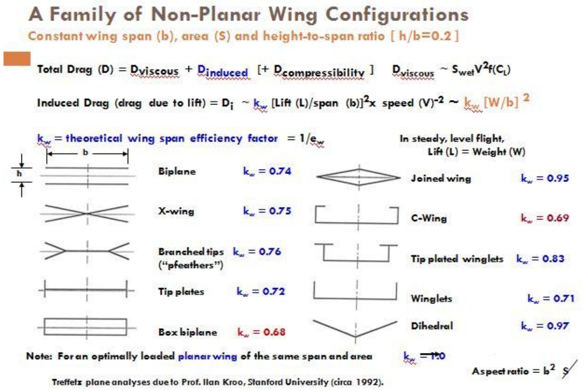


Fig 2. Family of Non Planar Wing

Finally some nonplanar concepts are motivated by other considerations including the possibility of improved high lift performance or desirable stability and control characteristics. In this paper we will consider several of these concepts, describe their potential advantages and difficulties, and explore some of the basic ideas that motivate many nonplanar wing designs[3].

Generally speaking, the requirements for minimum vortex-induced drag and minimum structural weight are diametrically opposed. In order to minimize the vortex-induced drag, the wing system must have either a large lateral, or, a large vertical dimension, usually leading to a heavy structure. Viscous effects and additional structural weight are two aspects of wing extension designs which must be carefully taken into consideration during the initial design phase[11].

C-Wing Configuration

The optimal loading of this lifting system is shown in the fig.1.6. The circulation of the main wing is carried onto the winglet so that the winglet is loaded inward. When the horizontal extension is added to the winglet, forming the "C" shape, the circulation is extended from the winglet as well, producing a surface that is loaded downward for minimum induced drag at fixed total lift. It is only when the lifting surface is extended to the centre line to form a box plane that the upper wing can efficiently carry an upload. This is because, as mentioned previously in connection with closed systems, we can superimpose a constant circulation ring on the closed system to redistribute the lift without changing the wake. This download on the C-wing horizontal surfaces affects structural weight and trim and the implications for aircraft configuration concepts was intriguing.[1]

However from this study it is very well understood that the C-wing possesses better aerodynamic performance than the conventional straight wing and it is meant for span constrained airplane, the structural characteristics remain important. Any aeroelastician would be struck by potential flutter penalties associated with large torsional inertias and coupling. Yet to avoid this, multiple ailerons may be used. C-wing also improves weight on wing which leads to a small improvement in total drag, still the L/D ratio is high, since tip vortices is reduced (induced drag). C-Wing also has some disadvantages over straight wing such manufacturing and maintenance complexity which may be given less importance due to its higher aerodynamic characteristics.

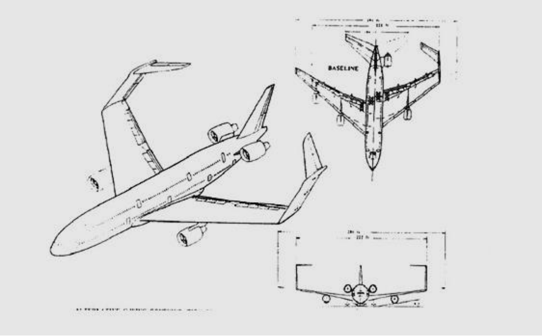


Fig.3. Design of C-wing

Advantages:

1. Reduced span or reduced vortex drag at fixed span
2. Efficient trim with short fuselage
3. Improved lateral handling (lower effective dihedral, reduced adverse yaw)
4. Potential for aero elastic control: prevent aileron reversal, active flutter control
5. Reduced tendency for pitch-up, control at high alpha
6. Reduced vertical tail height
7. Possible reduction in wake vortex strength

Disadvantages:

1. Details of emergency egress remain uncertain
2. Aerodynamics of thick inboard sections still an issue
3. Aeroelastics may be controllable but may need to be controlled.

II. COMPUTATIONAL FLUID DYNAMICS:

Computational Fluid Dynamics, popularly abbreviated as CFD, is a well-established and proven method or tool for simulating and studying fluid flows in the subjects involving fluid, heat and mass transfer in different conditions. CFD basically works by solving the governing equations of fluid dynamics, the Navier-Stokes equations, a coupled system of nonlinear partial differential equations, numerically as they are very difficult to solve analytically, with the help of specific
boundary conditions. Hence, CFD helps engineers find an approximate and nearly accurate solution to the governing equation for a range of fluid flow problems.

Computational fluid dynamics provides a qualitative and sometimes even quantitative prediction of fluid flow by means of mathematical modelling, numerical method and software tools. CFD analysis enables an engineer to compute the flow numerically in a ‘virtual flow laboratory’. The analysis consists of several steps such as: problem statement, mathematical modelling, mesh generation, space discretization, time discretization, iterative solver, simulation run, post processing, and verification.[12] The commercial CFD software solves the general transport equations using the finite volume method. Steady-state, transient, incompressible, compressible, inviscid, viscid, laminar, and turbulent flows can be solved with Fluent.

There are several CFD codes existing in the market which use different numerical techniques for solving the governing equations. Numerical methods used in CFD include

i. Finite-Difference Method
ii. Finite-Volume Method
iii. Finite Element Method

Commercially available CFD codes use finite-volume method or finite-element method.CFD is quite popular as an analysis tool in the industry due to its cost-effectiveness over physical testing methods such as wind tunnel tests. Also, in the specific case of wind tunnels, CFD score far better in terms of simulating the correct environment which is difficult to replicate in a wind tunnel and are also better in terms of accuracy as compared to wind tunnel test results.

A. *Modelling of commercial wing and c-wing:*

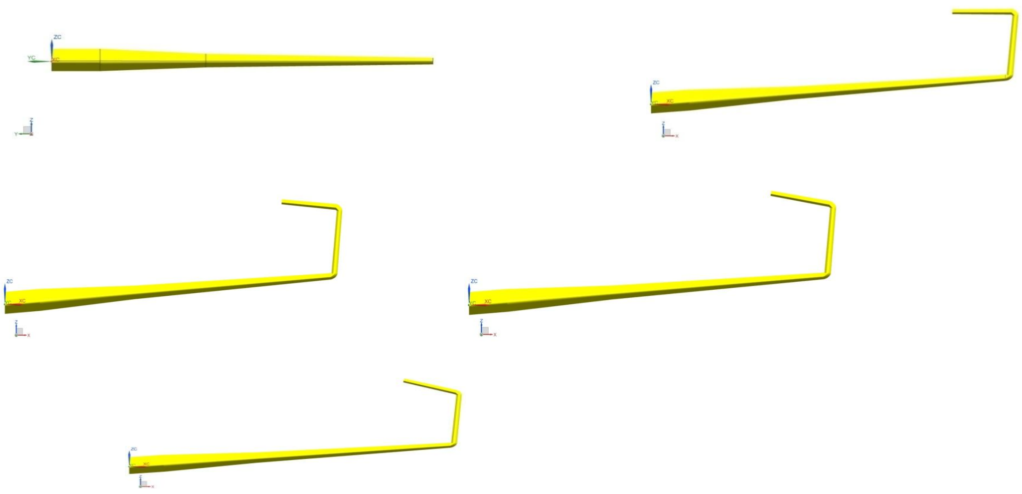


Fig.4. CAD Design of commercial wing and C-wing

Fig .4. shows the CAD models created for this analysis using NX-CAD. The geometric model that underwent CFD analysis is a 1:100 scale model of the original prototype whose dimensions are shown and also 3d-view projection also created using NX-CAD. It is essential to note here that only the size has been varied between the models and the rest of the geometry is unchanged.

- Height of the horizontal winglet = 0.2 * Semispan
- Length of the vertical winglet = 0.75 * height of horizontal winglet
- Wing profile is drawn with NACA 64-215(Supercritical airfoil)

Dimensions:

- Span of the wing(b) = 225mm
- Chord length(c) = 94mm @ x=0
- Chord length(c) = 53.7mm @ x=91mm
- Chord length(c) = 22.5mm @ x=225mm
- Height of vertical winglet(h) = 44.8mm
- Height of horizontal winglet(h) = 32.7mm
- Sweep angle of wing = 30º
- Dihedral angle of the wing = 5º

B. Hypermesh

Most FEA and CFD software do not have a great meshing algorithm; even when they have it, they do not allow good manual control on the mesh. The result is that after meshing a complicated geometry the mesh is lacking in quality, which means the analysis would either fail or give less than accurate answers. Additionally some elements (like beams, pretension, masses etc) are not easily created in FEM software. Hence a requirement of a good Finite Element (or Finite Volume in case of CFD) software is sometimes felt, especially in dealing with castings or large assemblies. HyperMesh made by Altair Inc is the most popular software in this regard. It is possible to create the entire analysis deck in an FE modelling software. Once done an analyst may just 'push' the deck to the solver However this is gradually falling out of favor in most OEMs as newer versions of solvers pack better meshing algorithms and thus reduce the time spent on the software and the investment on a package and training of the analyst. Newer packages are designed so as to help the designer himself perform the analysis and thus shave time off the product development cycle. This may not be the case however in a high Fidelity analysis in an aerospace or similar environment where speed is secondary and accuracy primary.

C. Meshing

- The pre-processing stage involved geometry setup and grid generation (meshing).
- The preprocessing program was done by Hypermesh.
- In this study, the half-symmetric model was imported into Hypermesh as a iges file.
- The computational domain (C Domain) was drawn around the 3-D geometry followed by the mesh generation using CFD Tetramesh .

- Boundary layer were given over the wing model with a thickness of 0.05m and the boundary layer growth of 1.20.

Meshing details are shown in Table.1and Fig 5 and 6 show the mesh models.

	Commercial wing	**C-wing**
Mesh	CFD Tetramesh	CFD Tetramesh
Boundary layer	5 layers	5 layers
Boundary layer thickness	0.05m	0.05m
Growth rate	1.2	1.2
No of elements	1002074	1278795
Surface area	$0.0277m^2$	$0.0319m^2$

Table.1. Meshing details of commercial and C-wing

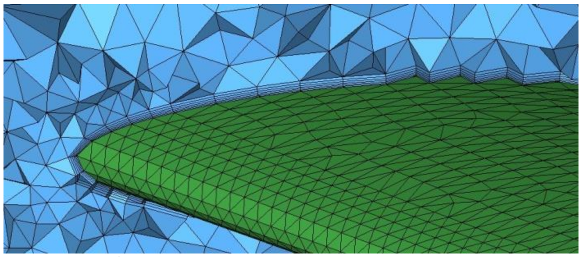


Fig.5. Meshing of commercial wing

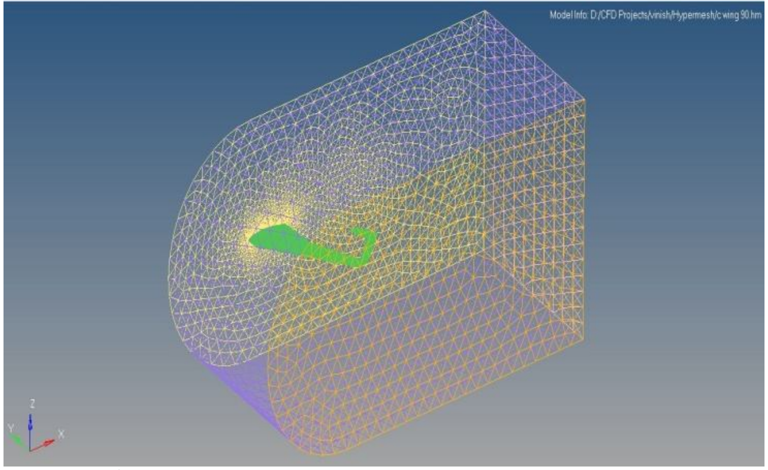


Fig.6. Meshing of C-wing

D. Boundary Condition

- The domain has 5 different types of boundaries surrounding it
- Inlet
- Outlet
- Side walls
- Symmetry wall
- Model surface

Inlet

- At the inlet, the boundary condition was set to a velocity of 220 m/s, normal to the inlet simulating the freestream. This would give a flow parallel to the longitudinal X-axis of the model.
- The velocity components are calculated for each angle of attack case as follows. The x-component of velocity is calculated by $x = u \cos\alpha$ and the y component of velocity is calculated by $y = u \sin\alpha$, where α is the angle of attack in degrees.

Outlet

- At the outlet boundary the total pressure was set to zero. This means that the air leaving the control volume will not experience any resistance nor will it be drawn out making its velocity higher than normal.

Side Walls

- The boundary was changed to a slip wall. The slip wall is a friction-less wall which only has the function to stop fluid passing through it. The reason this was used was because it would have the least effect on the flow around the model. This worked better although the solution did not converge as rapidly.

Symmetry wall

- The wall where the domain was split into two equal halves was set to slip wall, the same as the side walls.

Model Surface

- The surface of the model was set as a no-slip wall resulting in zero velocity at the node on the wall, letting a boundary layer form in the adjacent nodes as the computations progressed. This would give the right friction forces the flow exerts on the model.

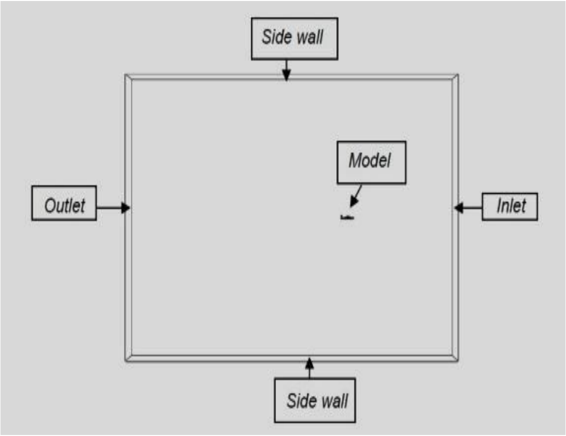


Fig 7. Types of boundaries

E. Solver Controls

After generating mesh it was feed to a solver to calculate the flow properties. Here the solver used was a FLUENT. ANSYS FLUENT is a state-of-the-art computer program for modelling fluid flow, heat transfer, and chemical reactions in complex geometries. ANSYS FLUENT uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstations and powerful computer servers. This architecture allows for efficient execution, interactive control, and complete flexibility between different types of machines or operating systems. ANSYS FLUENT provides complete mesh flexibility, including the ability to solve the flow problems using unstructured meshes that can be generated about complex geometries with relative ease. ANSYS FLUENT also allows to refine or coarsen the mesh based on the flow solution.[13]

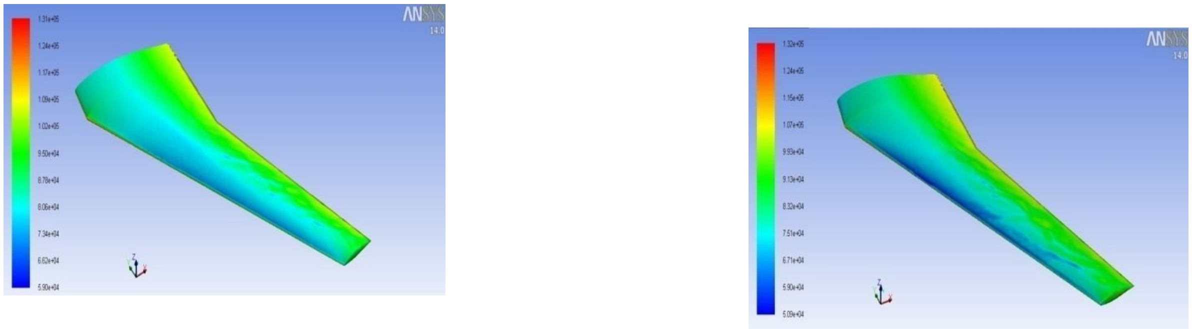
III. RESULTS AND DISCUSSION

A. *Cfd Analysis - Commercial Wing*

The CFD analysis of various wing models are shown below

PRESSURE CONTOUR

a) 0º angle of attack **b)2º angle of attack**



c)4º angle of attack **d)6º angle of attack**

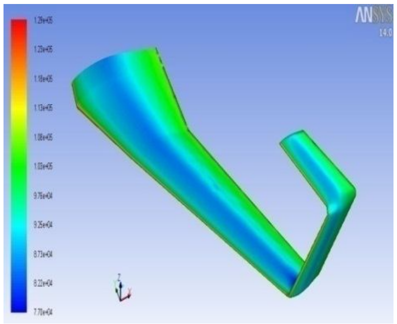
e)8º angle of attack **f)10º angle of attack**

Fig.8(a,b,c,d,e,f) show the pressure contours of commercial wing model at various angle of attack(0º,2º,4º,6º,8º,10º)

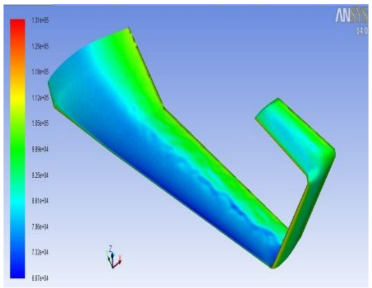
B. *C-WING WITH MOUNTING ANGLE 100º*

PRESSURE CONTOUR

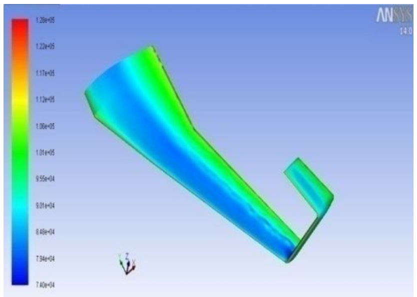
a)0º angle of attack



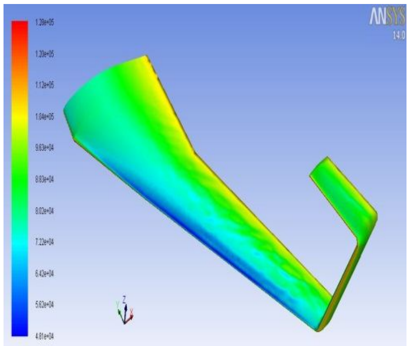
b)2º angle of attack



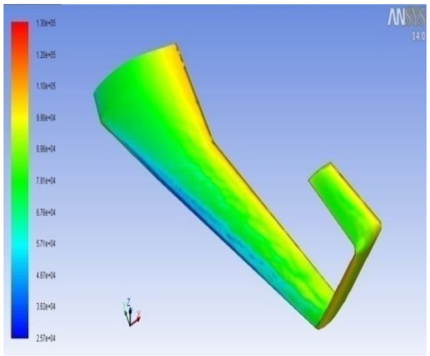
c)4º angle of attack

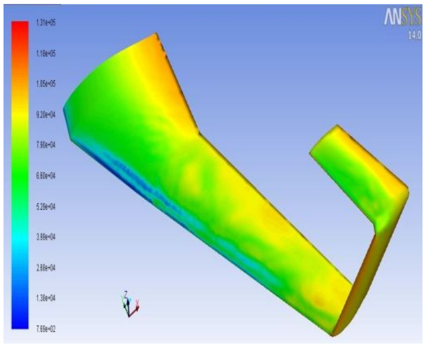


d)6º angle of attack

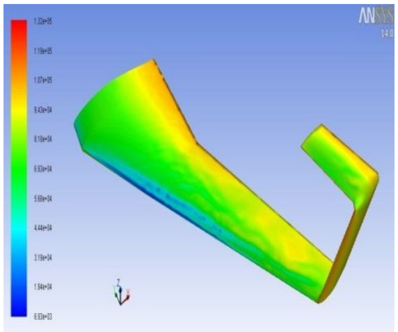


e)8º angle of attack





f)12º angle of attack



g)10º angle of attack

Fig.9. Pressure contour for C-wing with mounting angle 100º

Fig .9. (a,b,c,d,e,f,g) show the pressure contours of the C-wing with mounting angle 100º at various angle of attack(0º,2º,4º,6º,8º,12º,10º)

Similarly the CFD analysis has been carried out for the C wing with various mounting angles such as 90°, 95° and 105° and the results are tabulated in the table 2

C. Co-Efficient Of Lift And Drag

C_L/C_D Ratio for all angles:

The co-efficient of lift/co-efficient of drag ratio is the outcome of the observations made in the two preceding sections. It is observed from the Table .2. shows that the co-efficient of lift/co-efficient of drag ratio for all the configurations considered increases with an angle of attack to its maximum value and thereby it decreases with further increase in angle of attack.

Sl. No	Angle of attack(°)	Commercial wing	C-wing with mounting angle 90°	C-wing with mounting angle 95°	C-wing with mounting angle100°	C-wing with mounting angle 105°
1	0	23.53	22.06	22.09	22.19	21.88
2	2	42.9	44.55	45.39	46.32	35.69
3	4	29.39	29.58	30.49	32.27	26.18
4	6	17.75	19.17	20.04	20.07	16.64
5	8	14.47	15.13	15.25	15.27	12.59
6	10	9.85	11.40	11.57	11.65	8.81
7	12	6.49	9.01	9.24	9.25	6.92
8	14	4.49	6.91	6.94	7.12	5.47

Table.2. C_L/C_D Ratio

In particular it is observed that the maximum co-efficient of lift/co-efficient of drag ratio for all the configurations considered in the study falls in the range of 2 to 6 degrees of angle of attack. The commercial aircraft wing model gives a measured C_L/C_D ratio of 42.9 whereas the respective values of the C_L/C_D ratio for the different C-wing configurations are 44.55,45.39,46.32,35.69 for angle 90°,95°,100°,105° respectively at an angle of attack of 2°. The C_L/C_D ratio values for the angle of attack of 6° are 19.17,20.04,20.07,16.64 respectively for the different C-wing configurations.

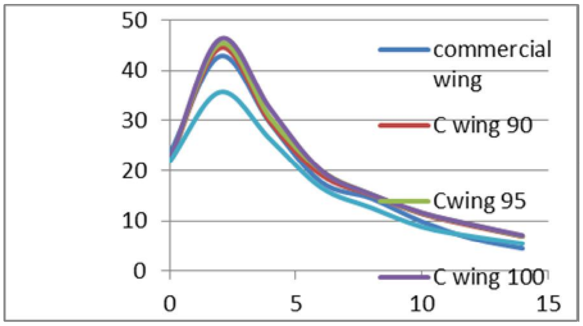


Fig. 10. Angle of attack Vs C_L/C_D

From the investigations, it is observed that at C-wing configuration of angle 90°,95°,100°,105° between the horizontal and vertical winglet provides the largest increase of C_L/C_D ratio ranging from 5% to7% increase .C-wing with angle 100° between the horizontal and vertical winglet has the better performance as compared to other configurations and it is giving the better C_L/C_D ratio.[4]

A. *Experimental testing:*

Experimental testing is an important tool required to validate all computational results. With current technology, vortex lattice methods and computational fluid dynamics remain attractive and relatively inexpensive tools to use for analysis of aircraft wings. However they still retain weakness with regard to the simulation of real life flow conditions, such as separation, turbulence and boundary layer growth. Experimental testing therefore remains a primary analysis tool for capturing such effects and along with flight testing and computational results is a fundamental tool in aircraft design. In itself, experimental wind tunnel testing also contains sources of error, with regard to the effects of the tunnel and model support on the model and airflow as well as potential calibration and Reynolds Number discrepancies.[5] Fig.11. shows the wind tunnel geometry specifications.

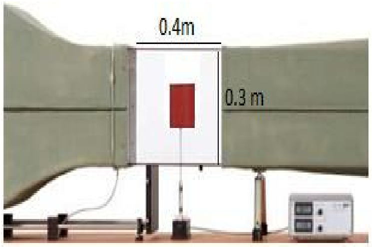


Fig.11. wind tunnel test area

Test conditions:

The measurements have been conducted in an open wind tunnel at aerodynamics laboratory, Hindusthan college of engineering and technology. This tunnel test section long is about 0.4m long and flow cross-section is approximately 0.3m×0.3m, interval of wind velocity upto 0.3mach and the contraction ratio of 9:1. The airfoil used in the present study is and academic NACA 64-215 profile (chord length c of 94mm). The experiment have been conducted at 30 m/s wind velocity (V) in tunnel.[14]

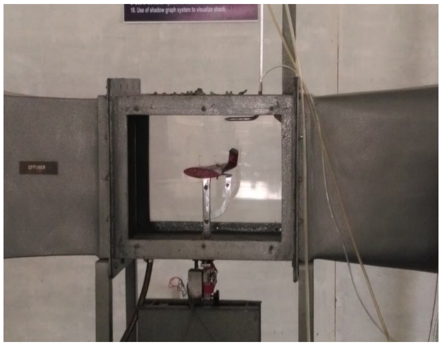


Fig.12.Wind Tunnel testing of C wing with 90°

E. *Validation of results*

C-WING WITH MOUNTING ANGLE 90° AT 30M/S

Angle of attack(°)	C_L	C_D
0	0.0021	0.000168
2	0.0035	0.000302
4	0.0064	0.000461

6	0.0075	0.000528

Table.3. C-Wing with mounting angle 90° at 30m/s

Table.3. shows the CFD analysis result for C-Wing with mounting angle 90° at 30m/s for validation of results with wind tunnel testing.

WIND TUNNEL TESTING

Angle of attack(°)	C_L	C_D
0	0.0021	0.00018
2	0.0036	0.00032
4	0.0068	0.00049
6	0.0081	0.00056

Table.4. C-wing with mounting angle 90° at 30m/s

The wind tunnel results for C-Wing with mounting angle 90° at 30m/s are tabulated in Table 4.

VALIDATION

Angle of attack(°)	**Experimental**		**Computational**		**Error percentage**	
	C_L	C_D	C_L	C_D	C_L	C_D
0	**0.0021**	**0.00018**	**0.0021**	**0.00016**	**7.2**	**6.6**
2	**0.0036**	**0.00032**	**0.0035**	**0.00030**	**6.3**	**7.9**
4	**0.0068**	**0.00049**	**0.0064**	**0.00046**	**6.5**	**6.1**
6	**0.0081**	**0.00056**	**0.0075**	**0.00052**	**7.4**	**6.2**

Table.5. Validation of results

Table.5. shows the comparison between experimental and computational result and it is seen that there is good agreement between them. The error percentage is estimated to be 6-8%.

IV. CONCLUSIONS

Following are the conclusions drawn from this investigation

i)Using C-wing configuration will increase lift force and reduce drag force and thereby increase the C_L/C_D ratio. By increasing the mounting angle, it further increases the C_L/C_D ratio and it is found to be optimum at 100°.

ii) The C-wing design is capable to reduce induced drag force and convert wing tip vortices to additional thrust which will save cost by reducing the usage of fuel, noise level reduction and increase the efficiency of the aircraft engine.

Hereby, this investigation provides a better understanding for the C-wing concept and its inclusion to the wing of aircraft wing model.

V. FUTURE WORK

In future, this single model wing design will be incorporated to the whole aircraft (Airbus A300 with various C-wing geometries) to perform computational and experimental investigation.

REFERENCES

[1] Ilan Kroo, *Highly Nonplanar Lifting Systems,* Stanford University John McMasters, Boeing Commercial Airplane Group and Stephen C. Smith, NASA Ames Research Center

[2] J Hoefling,A Spohr,D Schaefar, *Induced drag computations with wake model schemes for highly non planar lifting system* FH Aachen University of applied sciences,flight systems,flight guidance and control,52064 Aachen,Germany

[3**]** I. Kroo**,** *Nonplanar wing concepts for increased aircraft efficiency* ,Stanford University, U.S.A.

[4] Altab Hossain, Ataur Rahman, A.K.M. P. Iqbal, M. Ariffin, and M. Mazian, *Drag Analysis of an Aircraft Wing Model with and without Bird Feather like Winglet,* World Academy of Science, Vol:5 No:9, 2011.

[5] D. Smith*, M.H. Lowenberg*, D.P. Jones*, M.I. Friswell *computational and experimental analysis of the active morphing wing concept* **,**University of Bristol, Bristol, BS8 1TR,Swansea University, Singleton Park, Swansea, SA2 8PP, UK

[6] Mr. Darshak Bhuptani *Structural and modal analysis of A300 wing structure,* Indira Gandhi National Open University, New Delhi.

[7] Airbus S.A.S,A300-600 Airplane characteristics for airport planning,Technical data support an services,printed in france

[8] C.P.Van Dam,*Analysis of non-planar wing-tip-mounted lifting surfaces on low-speed airplanes,*GRANT NSG-1633,June 1983

[9] Adam Kosik,*The cfd simulation of the flow around the aircraft using solid works and fluent,*Evektor s.r.o.,Czech Republic

[10] P. K. Bahumanyam1, *Evaluation of Novel Wing Design for UAV,* University of Alabama in Huntsville, Huntsville, AL, USA

[11] Ulrich Kling,Corin Gologan, *Aeroelastic investigation of a self-trimming non-planar wing,*documentID:301274 Deutscher Luft-und,Germany.

[12] MD. Safayet Hossain, Muhammad Ferdous Raiyan, *A comparative flow analysis of naca 6409 and naca 4412 aerofoil*, International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308.

[13] Sangram Keshari Samal & P. K. Dash, *Reduction of Wingtip Vortex from Suction at Wingtip,* published by Canadian Center of Science and Education.

[14] İzzet Şahin and Adem Acir, *Numerical and Experimental Investigations of Lift and Drag Performances of NACA 0015 Wind Turbine Airfoil,* International Journal of Materials, Mechanics and Manufacturing, Vol. 3, No. 1, February 2015.

[15] A. Beechook, J. Wang, *Aerodynamic Analysis of Variable Cant Angle Winglets for Improved Aircraft Performance,* Brunel University,London, UK, 13-14 September 2013.

[16]. C. Suresh, Ramesh. K, Paramaguru V, *Aerodynamic performance analysis of a non-planarC-wing using CFD,* Aerospace science and Technology 40 (2015)56-61, October 2014