Aero Dynamic Analysis of Multi Winglets in Light Weight Aircraft

J. Mathan[#1],L.Ashwin[#2], P.Bharath[#3,]P.Dharani Shankar[#4],P.V.Jackson[#5]
[#1]*Assistant Professor & Mechanical Engineering & KSRIET*
[#2,3,4,5]*Final Year Student & Mechanical Engineering & KSRIET*
Tiruchengode,Namakkal(DT),Tamilnadu

Abstract

An analysis of multi-winglets as a device for reducing induced drag in low speed aircraft is carried out, based on experimental investigations of a wing-body half model at Re = 4•105. Winglet is a lift augmenting device which is attached at the wing tip of an aircraft. A Winglets are used to improve the aerodynamic efficiency of an aircraft by lowering the formation of an Induced Drag which is caused by the wingtip vortices. Numerical studies have been carried out to investigate the best aerodynamic performance of a subsonic aircraft wing at various cant angles of winglets. A baseline and six other different multi winglets configurations were tested. The device led to 32% improvement in the Oswald efficiency factor, representing an increase of 7% in the maximum aerodynamic efficiency. Improvements of 12% in the maximum rate of climb and 7% in the maximum range were also obtained. The pressure distribution was measured to verify global and local effects of the multi winglets, showing only a small influence of the device on the wing loading. Structural investigations were also carried out, as well as wake surveys using a seven hole Pitot probe that indicated significant changes in the flow field near the wingtip. The design process is carried out in CATIA. Discretization and the CFD simulation has been carried out through fluent, and the Post-processing results are obtained using ansys.

Keywords: *lift and drag force, laminar and turbulent flow, CFDanalysis, angle of attack..*

I. INTRODUCTION

Induced drag is caused by the wingtip vortex, an unavoidable collateral effect of lift generation in a finite wing. It has been proven that modifications in the wingtip or the use of wingtip devices can minimize the induced drag expressively. Extensive research was conducted with the objective of studying these devices, as well as proposing new designs and approaches. In this context, the present work investigates the potential use of multi-winglets to decrease the induced drag in a light aircraft, enhancing the aerodynamic efficiency and the performance. Modifications in the wingtip can either move the vortices away in relation to the aircraft longitudinal axis or reduce their intensity [1]. Some of these devices such as winglets [2], tip-sails [3, 4, 5] and multi-winglets [6] take energy from the spiraling air flow in this region to create additional traction. This makes possible to achieve expressive gains on efficiency. Whitcomb [2], for example, shows that winglets could increase wing efficiency in 9% and decrease the induced dragin20%. Some devices also break up the vortices into several parts, each one with less intensity. This facilitates their dispersion, an important factor to decrease the time interval between takeoff and landings at large airports [7]. A comparison of the wingtip devices [1] shows that winglets have higher aerodynamic benefits upto Mach1.0, but may create structural problems due to the bending moment increase at the wing root. Tip-sails have the same drag reduction at low lift conditions, with a lower bending moment at the wing root. In general aviation, research on wingtip devices was carried out for sailplanes, even though their wings have a large aspect ratio.

A. Introduction to wings

Wings are airfoils that, when moved rapidly through the air, create lift. They are built in many shapes and sizes. Wing design can vary to provide certain desirable flight characteristics. Control at various operating speeds, the amount of lift generated, balance, and stability all change as the shape of the wing is altered. Both the leading edge and the trailing edge of the wing may be straight or curved, or one edge maybe straight and the other curved. One or both edges may be tapered so that the wing is narrower at the tip than at the root where it joins the fuselage. The wingtip may be square, rounded, or even pointed. Shows a number of typical wing leading and trailing edge shapes. The wings of an aircraft can be attached to the fuselage at the top, mid-fuselage, or at the bottom. They may extend perpendicular to the horizontal plain of the fuselage or can angle up or down slightly. This angle is known as the wing dihedral. The dihedral angle affects the lateral stability of the aircraft. Shows some common wing attach points and dihedral angle.

1. Types Of Wings

- Non planar wing or closed wing
- Box wing
- Annular (cylindrical)

- Joined wing
- Annular wing (planar)
    - Flat
    - Rhomboidal wing

2. Wing Structure

The wings of an aircraft are designed to lift it into the air. Their particular design for any given aircraft depends on a number of factors, such as size, weight, use of the aircraft, desired speed in flight and at landing, and desired rate of climb. The wings of aircraft are designated left and right, corresponding to the left and right sides of the operator when seated in the cockpit. Often wings are of full cantilever design.

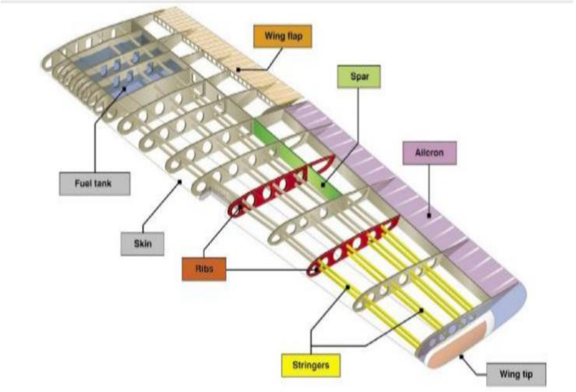


Fig. 1.1 simple aircraft wing structure

This means they are built so that no external bracing is needed. They are supported internally by structural members assisted by the skin of the aircraft. Other aircraft wings use external struts or wires to assist in supporting the wing and carrying the aerodynamic and landing loads.

3. Variable configurations multi-winglets

The wing tip device is a variable configuration multi-winglets with three tip sails. Those are attached to a movable mechanism that allows the adjustment of the cant angle and incidence for each sail independently. In order to avoid undesirable configuration change during experiments, a locking mechanism was built. All pieces of the wingtip device were manufactured in polyamide

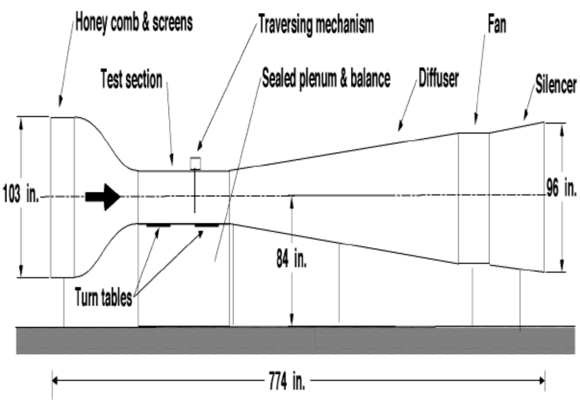


Fig.1.2.1 Model prepared for the wing tunnel testing

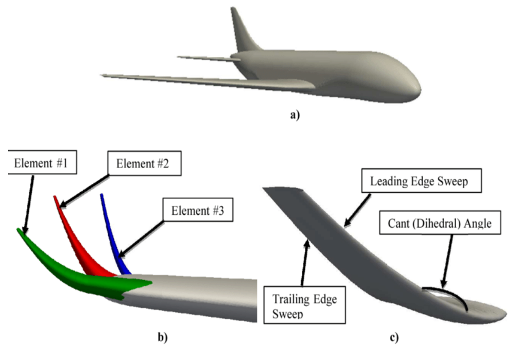


Fig. 1.2.2 Multi-winglets details

The initial estimation of the flow field was performed with CFD calculations. Six configurations are compared with the baseline (no sails configuration). The cant angles combinations are listed in Tab. 1 and they are based on the best results available in [10]. Trips for forcing transition were not used during most of the tests, as the objective of the present work is the study of the lift dependent drag. One case with forced transition was evaluated and compared to the un tripped wing case. The fully turbulent case had slight lower lift co efficients, however, no changes in the stall characteristics due to the forced transition were detected.

Wingtip devices are intended to improve the efficiency of fixed-wing aircraft by reducing drag.[1] Although there are several types of wing tip device, which function in different manners, their intended effect is always to reduce an aircraft's drag by partial recovery of the tip vortex energy. Wingtip devices can also improve aircraft handling characteristics and enhance safety for following aircraft. Such devices increase the effective aspect ratio of a wing without greatly increasing the wingspan. Extending the span would lower lift-induced drag, but would increase parasitic drag and would require boosting the strength and weight of the wing. At some point, there is no net benefit from further increased span**.** At some point, there is no net benefit from further increased span

B. Methodology followed in the project

A comparison between the results obtained based on existing material and the results obtained from the ANSYS. Work bench has been carried out.

In conventional approach conception ideas are converted into sketches or engineering drawing. With the help of this drawings the prototypes i.e. product which looks same as that of final product are made. It is launched in the market after testing of prototype which gives acceptable results. The thing is, product is launched after doing many practical testing and many trial and error procedures which consumes more time and cost too.

In CAE approach some steps are same as that of conventional method. Here also ideas, concepts are converted into engineering drawing, but it is then modelled on computer. Geometric model of product is made using solid work software like CAD which enables better visualization of simple as well as complex models. These models then further used for computerized analysis by using different CAE tools (FEA / CFD software's) depending upon the application before the prototype is been made to check whether the components are going to work according to its intended function. After that once appropriate results are obtained the final practical testing is carried out.

C. Objective

The present work investigates the potential use of multi-winglets to decrease the induced drag in a light aircraft, enhancing the aerodynamic efficiency and the performance.

II. MATERIAL SELECTION

The Winglet NACA 4412 air foil is used to design three types of winglets at different Cant angles. As in most air foil design efforts, the goal of the winglet

Airfoil design is to generate the lift required with the lowest possible drag. The winglets are designed at four different Cant angles i.e. 0°, 45°, 60°, 90°.The images of the these winglets designed by CATIA V5.The material is used for this winglet is Fiber Glass Re-enforced Composite Material (FRCM) and wing material is wood.

III. MODELLING

Modelling is the testing of design ideas to see if they contribute to a fit-for-purpose technological outcome. There are two types of modelling.They are,

- Functional modelling
- Protype Modelling

A. Introduction To CAD/CAM

CAD/CAM is a term which means computer-aided design and computer-aided manufacturing. It is the technology concerned with the use of digital computers to perform certain functions in design and production. This technology is moving in the direction of greater integration of design and manufacturing, two activities which have traditionally been treated as district and separate functions in a production firm. Ultimately, CAD/CAM will provide the technology base for the computer-integrated factory of the future.

Computer – aided design (CAD) can be defined as the use of computer systems to assist in the creation, modification, analysis, or optimization of a design. The computer systems consist of the hardware and software to perform the specialized design functions required by the user firm. The CAD hardware typically includes the computer, one or more graphics display terminals, keyboards, and other peripheral equipment. The CAD software consists of the computer programs to implement computer graphics on the system plus application programs to facilitate the engineering functions of the user company.

Examples of these application programs include stress-strain analysis of components, dynamic response of mechanisms, heat-transfer calculations, and numerical control part programming. Computer-aided manufacturing (CAM) can be defined as the use of computer systems to plan, manage, and control the operations of manufacturing plant through either direct or indirect computer interface with the plant's production resources.

IV. DESIGN PROCESS

The process of designing is characterized by six identifiable steps or phase

1. Recognition of need
2. Definition of problem
   1. Analysis and optimization
   2. Evaluation
   3. Presentation
   4. Synthesis

A. Application of computer for design

The various design-related tasks which are performed by a modern computer-aided design system can be grouped into four functional areas:

1. Geometric modelling
2. Engineering analysis
3. Design review and evaluation

1. Geometric Modelling

In computer-aided design, geometric modeling is concerned with the computer- compatible mathematical description of the geometry of an object. The mathematical description allows the image of the object to be displayed and manipulated on a graphics terminal through signals from the CPU of the CAD system. The software that provides geometric modeling capabilities must be designed for efficient use both by the computer and the human designer.

There are several different methods of representing the object in geometric modeling. The basic form uses wire frames to represent the object. Wire frame geometric modeling is classified into three types, depending on the capabilities of the interactive computer graphics system.

2. Engineering Analysis

CAD/CAM systems often include or can be interfaced to engineering analysis software which can be called to operate on the current design model. Examples of this type are

1. Analysis of mass properties

2. Finite element analysis

The analysis may involve stress –strain calculations, heat-transfer computations, or the use of differential equations to describe the dynamic behavior of the system being designed.

V. ANALYSIS

A. Introduction to FEM

In mathematics, the finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It uses variation methods (the calculus of variations) to minimize an error function and produce a stable solution. Analogous to the idea that connecting many tiny straight lines can approximate a larger circle, FEM encompasses all the methods for connecting many simple element equations over many small sub domains, named finite elements, to approximate a more complex equation over a larger domain.

The subdivision of a whole domain into simpler parts has several advantages:

- Accurate representation of complex geometry
- Inclusion of dissimilar material properties
- Easy representation of the total solution
- Capture of local effects.

FEM is best understood from its practical application, known as finite element analysis (FEA). FEA as applied in engineering is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm.

In applying FEA, the complex problem is usually a physical system with the underlying physics such as the Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations expressed in either PDE or integral equations, while the divided small elements of the complex problem represent different areas in the physical system.

FEA is a good choice for analyzing problems over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation).

Another example would be in numerical weather prediction, where it is more important to have accurate predictions over developing highly nonlinear phenomena (such as tropical cyclones in the atmosphere, or eddies in the ocean) rather than relatively calm areas.

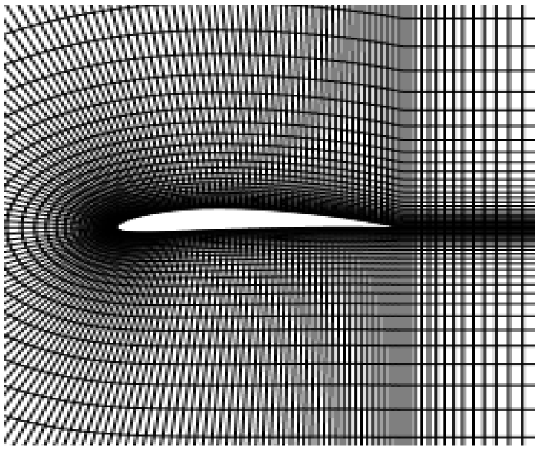


Fig 5 Meshing

FEM mesh created by an analyst prior to finding a solution to a magnetic problem using FEM software. Colours indicate that the analyst has set material properties for each zone, in this case a conducting wire coil in orange; a ferromagnetic component (perhaps iron) in light blue; and air in grey. Although the geometry may seem simple, it would be very challenging to calculate the magnetic field for this setup without FEM software, using equations alone.

FEM solution to the problem at left, involving a cylindrically shaped magnetic shield. The ferromagnetic cylindrical part is shielding the area inside the cylinder by diverting the magnetic field created by the coil (rectangular area on the right). The colour represents the amplitude of the magnetic flux density, as indicated by the scale in the inset legend, red being high amplitude. The area inside the cylinder is low amplitude (dark blue, with widely spaced lines of magnetic flux), which suggests that the shield is performing as it was designed to.

Finite element analysis (FEA) involves solution of engineering problems using computers. Engineering structures that have complex geometry and loads, are either very difficult to analyze or have no theoretical solution. However, in FEA, a structure of this type can be easily analyzed. Commercial FEA programs, written so that a user can solve a complex engineering problem without knowing the governing equations or the mathematics, the user is required only to know the geometry of the structure and its boundary conditions. FEA software provides a complete solution including deflections, stresses and reactions.

FEA is divided in 3 steps

B. Pre-processing

Using a CAD program the structure is modeled. A model consists of several elements that collectively represent the entire structure. The geometry of the structure, the constraints, loads and mechanical properties of the structure are defined. Thus, in pre-processing, the entire structure is completely defined by the geometric model. The structure represented by nodes and elements is called mesh.

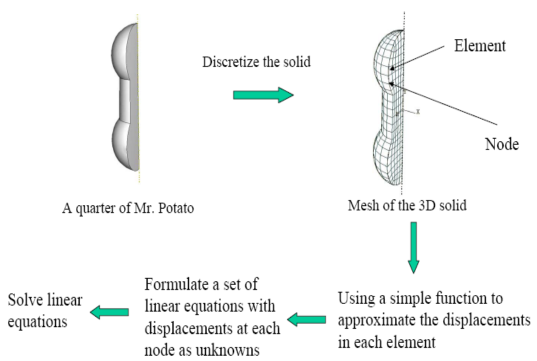
C. Solution

This phase can be performed in the Model Solution task of the simulation application, or in an equivalent external finite element solver. Model Solution can solve for linear and nonlinear static, dynamics, buckling, heat transfer, and potential flow analysis problems.

D. Post-processing

CAD program is utilized to manipulate the data for generating deflected shape of the structure, creating stress plots, animation. Graphical representation of the results is useful in understanding behaviour of the structure.

VI. HOW FEA WORKS



A. Computational fluid dynamics

Computational Fluid Dynamics (CFD) is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.)

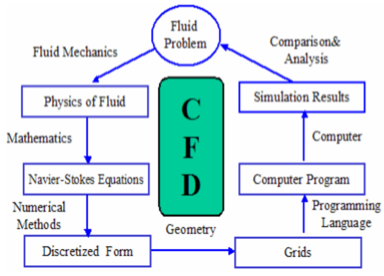


Fig 6.1 Process of CFD

Firstly, we have a fluid problem. To solve this problem, we should know the physical properties of fluid by using Fluid Mechanics. Then we can use mathematical equations to describe these physical properties. This is Navier-Stokes Equation and it is the governing equation of CFD. As the Navier-Stokes Equation is analytical, human can understand it and solve them on a piece of paper. But if we want to solve this equation by computer, we have to translate it to the discretized form. The translators are numerical discretization methods, such as Finite Difference, Finite Element,

Finite Volume methods. Consequently, we also need to divide our whole problem domain into many small parts because our discretization is based on them. Then, we can write programs to solve them. The typical languages are Fortran and C. Normally the programs are run on workstations or supercomputers. At the end, we can get our simulation results. We can compare and analyze the simulation results with experiments and the real problem. If the results are not sufficient to solve the problem, we have to repeat the process until find satisfied solution. This is the process of CFD.

B. Analysis using ansys software

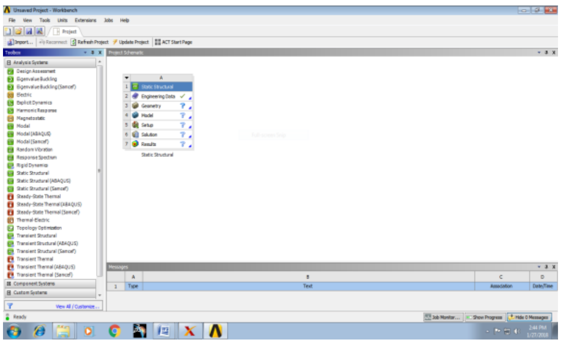


Fig 6.2 ANSYS Work bench

VII. RESULTS AND DISCUSSION

A. Wind tunnel corrections

Wind tunnel wall corrections and stand-off geom-etry were computed with CFD simulations of the baseline configuration and the results were com-pared with the respective experimental data.

The CFD computations were performed with ANSYS CFX, using the shear stress transport turbulence model (SST) and the g q transition model. Mesh generation was done with ANSYS ICEM CFD, using unstructured grids based on the real model, which geometry was confirmed with a high precision three-dimensional measur-ing arm.

CATIA V5 was used to generate the geome-try from the model surface scan data. Mesh con-vergence and validation studies were performed using the wind tunnel boundary conditions in the simulations. Once validated, the numerical model was evaluated for the wind tunnel condi-tions, considering the walls.

The results of both simulations were com-pared, giving relations of the aerodynamic coeffi-cients in the wind tunnel and in flight conditions, which were then applied to define corrections for typical analytical relations.

B. Aerodynamic characteristics

1. Lift and drag influence

Data obtained with the measured aerodynamic forces shows a significant increase in the lift curve slope when compared with the basic wing, achieving higher lift coefficients for most angles of attack. This can be explained mainly by the wing loading increase near the tip in addition to the lift of the each winglet itself, even with their small area. The lift curve slope was increased from 4:8 to 5:3 in configuration 5, which represents 11%.

VIII. REPORTS OF FLOW SIMULATION FOR AIRCRAFT

A. Design Of Existing Model Aircraft

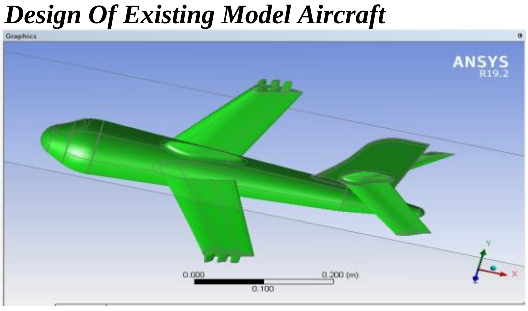


Fig 8.1

B. Analysis of existing model aircraft

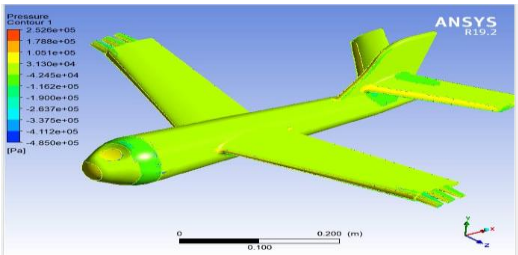


Fig 8.2

C. Winglets Pressure Contour

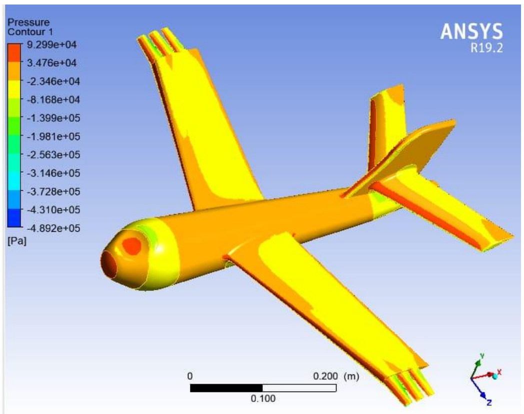


Fig 8.3

D. Meshing Of Winglets

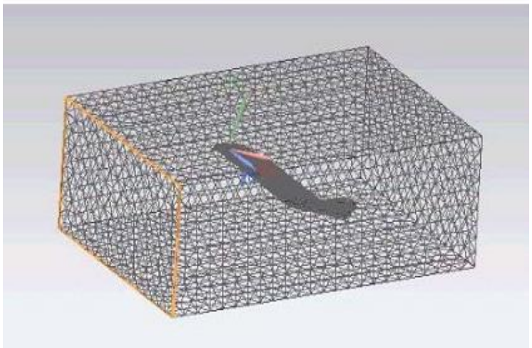


Fig 8.4

E. Meshing

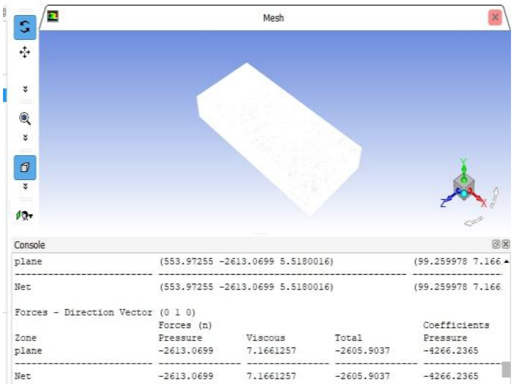


Fig 8.5

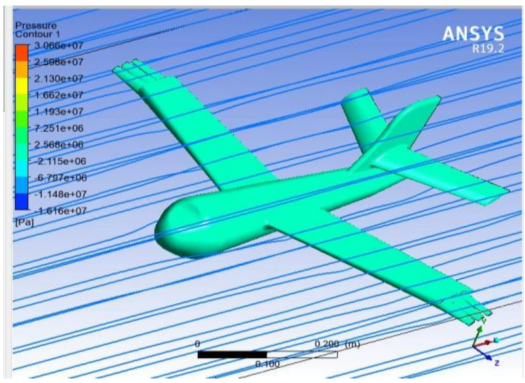


Fig 8.6

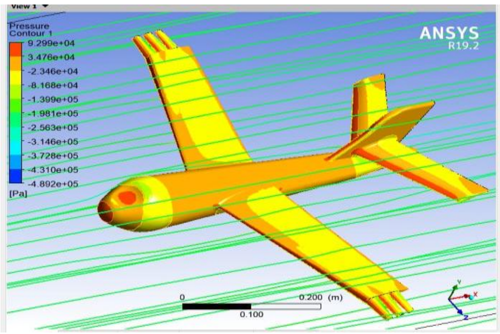


Fig 8.7

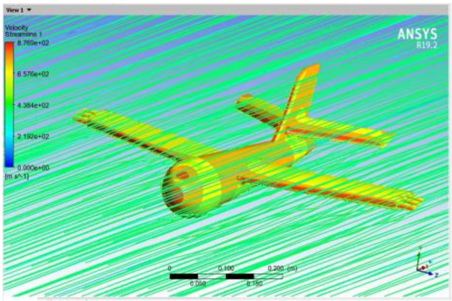


Fig 8.9

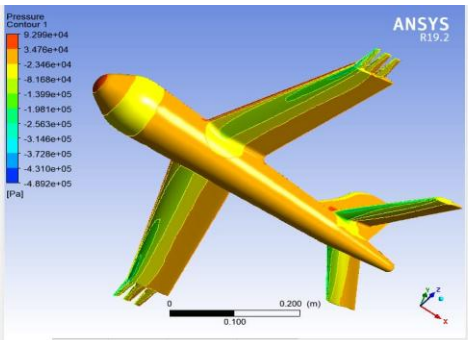


Fig 8.8

F. Meshing Of Wings

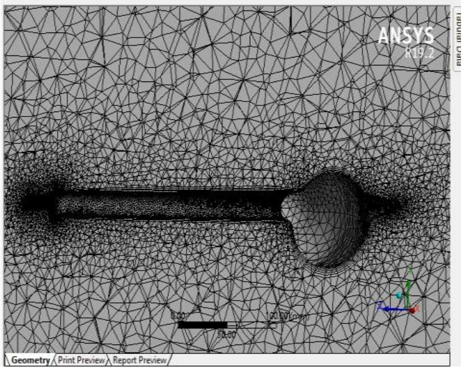


Fig 8.10

F. Graph

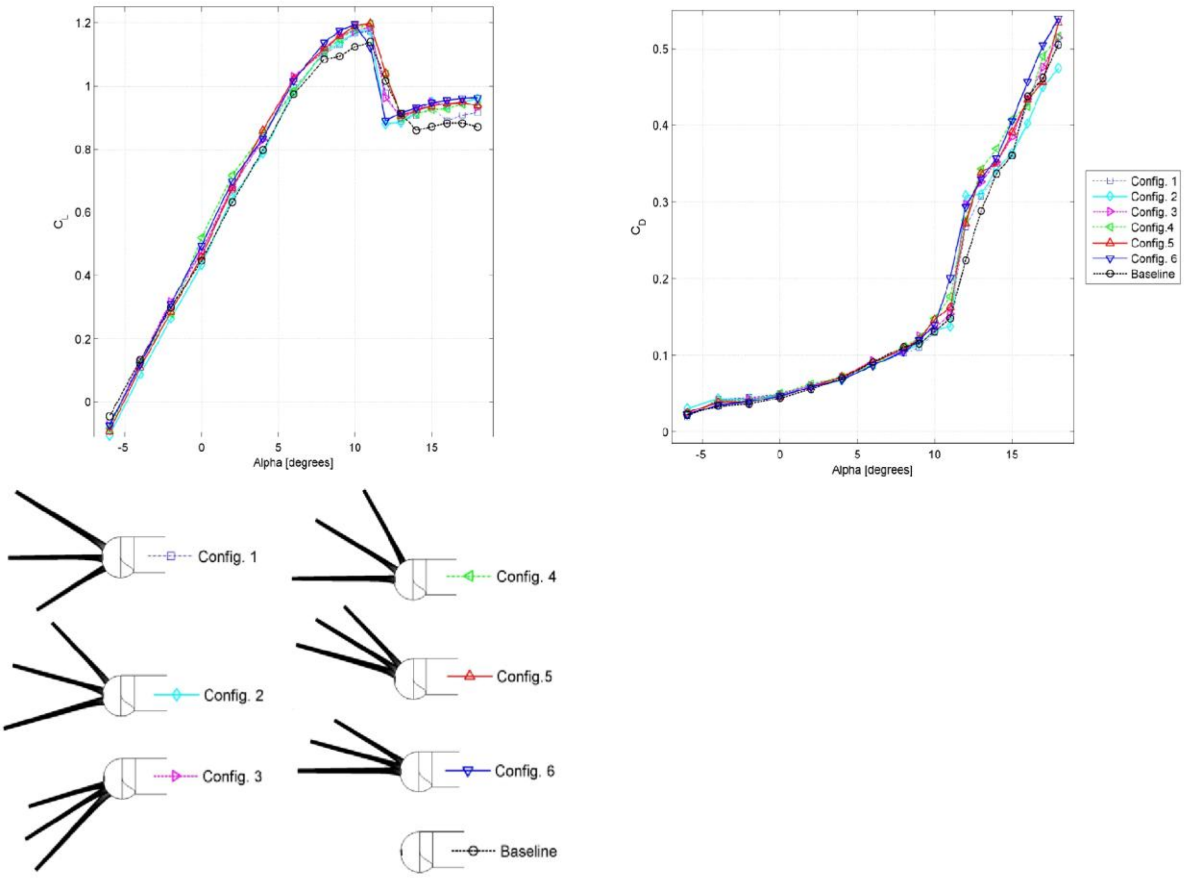


Fig 8.11 Graph for Lift coefficient and the studied configurations

The maximum lift coefficient raised from 1:14 to 1:20 in configuration.The effects of the multi-winglets on drag at most of angles of attack are of small magnitude. It can be seen that for angles of attack up to 2 , the device produces a slightly more drag than the basic wing. For higher angles, as in Fig. 4, the reductions of the induced drag become more ex-pressive and the total drag is reduced until the stall. These characteristics are consequence of the additional zero lift drag caused by the sails and the induced drag reduction, which is the prominent in most conditions.

The drag polar shows that the effi-ciency increase is due to greater lift and slightly lower drag, as the multi-winglets create a scattered vortex system, besides weaker and further from the wing tip, with lower influence over the wing. The induced drag reduction is better under-stood in Fig. 6, which shows that configurations with multi-winglets have a lower slope in the total drag as a function of C_L^2.

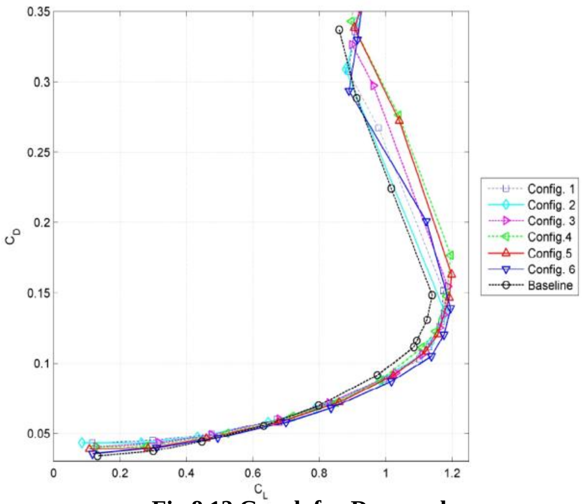


Fig 8.12 Graph for Drag polars

The reductions on the lift dependent drag are confirmed by the significant increases of up to32% the Oswald efficiency factor.Despite the clear induced drag reduction ,there is an undesirable increase in the zero lift drag. Roughly,wings with the device are more efficient only for lift

IX. CONCLUSION

- In this project aerodynamic characteristics (lift and drag) of the wing and winglet model with NACA 4412 air foil have been presented.
- In Blended winglet at 90 degrees Cant angle and at the same time the drag decreases more for the aircraft model with blended winglet.
- The Bird like winglet increases the lift more efficiently there by reducing the drag at lower speeds.

REFERENCE

[1] Richard Whitcomb, "Methods for Reducing Aerodynamic Drag," NASA Conference Publication 2211, Proceedings of Dryden Symposium, Edwards, California, 16 September 1981

[2] Richard Whitcomb, "A Design Approach and Selected Wind-Tunnel Results at High Subsonic Speeds for Wing-Tip Mounted Winglets," NASA TN D-8260, July 1976

[3] Yates, John E., and Donaldson, Coleman duP., Fundamental Study Of Drag And An Assessment Of Conventional Drag-Due-To-Lift Reduction Devices, NASA Contract Rep 4004, Sep 1986

[4] Ilan Kroo, John McMasters, and Stephen C. Smith, Highly Nonplanar Lifting Systems, Presented at: Transportation Beyond 2000: Technologies Needed for Engineering Design, NASA Langley Research Center, September 26-28, 1995

[5] Gall, Peter D., and Smith, Hubert C., Aerodynamic Characteristics of Biplanes with Winglets, Journal of Aircraft v 24 n 8 Aug 1987 p 518-522

[6] Louis B. Gratzer, Spiroid-Tipped Wing, U. S. Patent 5,102,068, 7 April 1992

[7] Ruhlin, Charles L., Bhatia, Kumar G., and Nagaraja, K. S., Effects Of Winglet On Transonic Flutter Characteristics Of A Cantilevered Twin-Engine Transport Wing Model. NASA Technical Paper 2627 Dec 1986

[8] Meyer, Robert R. Jr., and Covel, Peter F., Effects Of Winglets On A First-Generation Jet Transport Wing: VII—Side slip Effects On Winglet Loads And Selected Wing Loads At Subsonic Speeds For A Full-Span Model. NASA Technical Paper 2619 Sep 1986 58p

[9] Satran, Dale R.,Wind Tunnel Investigation of the Flight Characteristics of a Canard General aviation Airplane Configuration. NASA Technical Paper 2623 Oct 1986 59p

[10] Reginald V. French, Vortex Reducing Wing Tip, U. S. Patent 4,108,403, 22 August 1978

[11] Joe Clark, President, Aviation Partners, Inc., personal communication, 12 July 1999