FLOW SIMULATION FOR AIRCRAFT USING COMPUTATIONAL FLUID DYNAMICS

Mr. D. Ramkumar[1], M. Alexander[2], M. Balaji[3], K.N. Bharathi[4], R.M. Elanthendral[5]

[1]Assistant Professor, [2,3,4,5] B.E Scholar in Department of Mechanical Engineering, KSR Institute for Engineering and Technology, Tamilnadu, India.

Abstract - *In this project we describe the complete process of modelling and simulation of computational fluid dynamics (CFD) problems that occur in engineering field. Our project aims to perform a CFD analysis on an aircraft model using CFX solver package. We are going to change the angle of aircraft wing for better take off and safe landing on earth, also we are going analyse on CFD FLUENT. While performing the simulations, meshing techniques, pre-processing & post-processing sections and evaluation of a simulation is being performed . The CFD simulations were performed on the computational model of the twin turboprop aircraft EV-55 Outback. We are planning to make two model comparison between ANSYS Fluent and measurements in a wind tunnel. By using solver based on the semi-implicit method for pressure linked equation. The computations were performed for different model settings and computational grids, that we considered laminar and turbulent flow and several combinations of the angle of attack and inlet velocity. Coefficient of lift and drag were also recorded as a user input data. Also these values were compared by running two different simulations with one change of input parameter i.e, angle of attack and inlet velocity.*

Key Words: Lift and drag force, Laminar and Turbulent flow, CFD analysis, Angle of attack.

1. INTRODUCTION

The aircraft is a well-designed structured body basically used for moving in air by gaining the strength from atmosphere as it moves against the force of the gravity. In earlier days aircraft were seen as transportation medium but with the increase in new ideas and innovations, aircraft were introduced into the military power, thus we could see aircraft flying during the world wars. As development in the globalization continued, the development in many fields also enhanced. With these development we can see many new innovation in designs of aircraft to make it as innovative as possible .One such innovation were, the considerations of properties such as the static and dynamic lift to counter the gravitational force in order to fly the aircraft at great highest and also to withstand against the force of the trust .The aerodynamics force is found to be a central idea behind the design of aircraft.

The CFD simulations were performed on the computational model of the twin turboprop aircraft EV-55 Outback. Thus the present generation aircrafts include a lot of multidisciplinary designs to achieve the critical mission requirements such as endurances, maneuverability, fuel consumption, payload capacity, noise emission etc., when compared to the older once.

Furthermore, we present a way of analysing the results and some of the interesting outputs of the simulations and following analysis. We focus mainly on the simulation of the airflow around the aircraft. The fluid flow simulations are obtained from CFD software package. The important part is the preparation of the model with the software ANSYS

1.1 INTRODUCTION TO AIRCRAFT

An aircraft is a machine that is able to fly by gaining support from the air. It counters the force of gravity by using either static lift or by using the dynamic lift of an airfoil, or in a few cases the downward thrust from jet engines. Common examples of aircraft include airplanes, helicopters, airships (including blimps), gliders, and hot air balloons.

The human activity that surrounds aircraft is called aviation. The science of aviation, including designing and building aircraft, is called aeronautics.

Fig.1.1 The twin turboprop aircraft EV-55 outback.

1.2 DEVELOPMENT OF JET AIRCRAFT

The first practical jet aircraft was the German Heinkel He 178, which was tested in 1939. In 1943, the Messerschmitt Me 262, the first operational jet fighter aircraft, went into service in the German Luftwaffe. In October 1947, the Bell X-1 was the first aircraft to exceed the speed of sound.

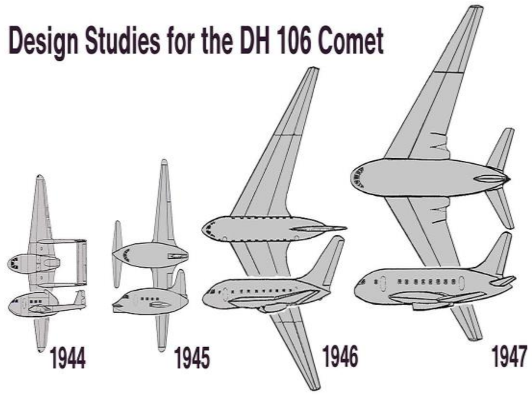

Fig. 1.2. Development of jet aircraft

The first jet airliner, the de Havilland Comet, was introduced in 1952. The Boeing 707, the first widely successful commercial jet, was in commercial service for more than 50 years, from 1958 to 2010. The Boeing 747 was the world's biggest passenger aircraft from 1970 until it was surpassed by the Airbus A380 in 2005.

1.3 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 modeled 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.

1.4 OBJECTIVE

In the present paper analyzed delta wing conceptual aircraft model on the parameters of speed at sub sonic speed, angle of attack, drag force, lift force generated, and stall angle and turbulences. The result obtained would determine its usability for fighter jets, commercial aircrafts and UAV drones. The technique used to analyze the problem is Computational Fluid Dynamics and software is ANSYS CFD. Our main aim is to increase lift force with decrease in drag force with help of vertex generator on the wing.

2. MODELING

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

- Functional modeling
- Prototype Modeling

2.1 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.

3. DESIGN PROCESS

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

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 03 | Mar 2019 www.irjet.net p-ISSN: 2395-0072

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

3.1 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 modeling
2. Engineering analysis
3. Design review and evaluation

3.1.1 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.

3.1.2 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.

4. ANALYSIS

4.1 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.

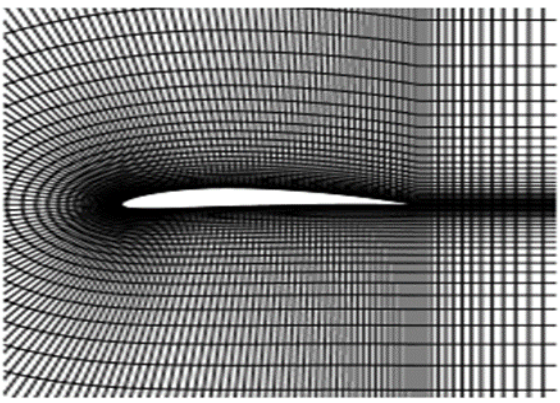

Fig 4.1 Meshing

FEM mesh created by an analyst prior to finding a solution to a magnetic problem using FEM software. Color 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 color 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.

4.2 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.

4.3 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.

4.4 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 behavior of the structure.

5. 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.)

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.

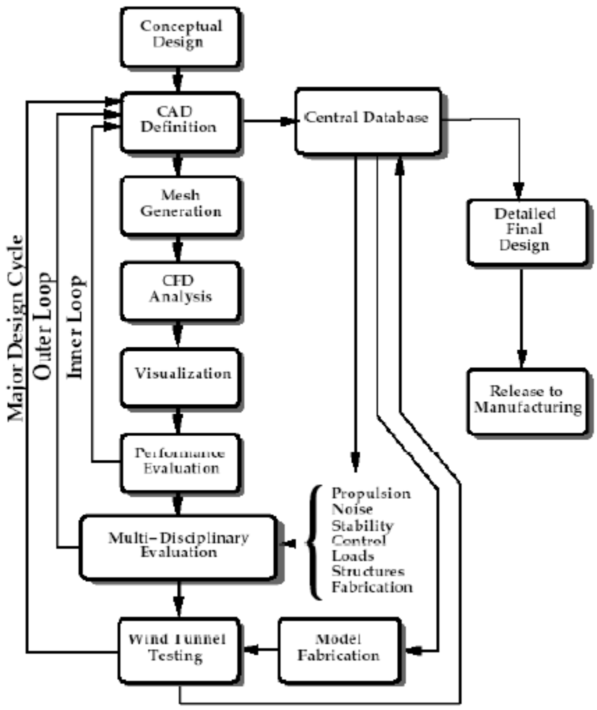

Fig 5.1 Overall Preliminary Design.

6. REPORTS OF FLOW SIMULATION FOR AIRCRAFT

6.1 DESIGN OF EXISTING MODEL AIRCRAFT

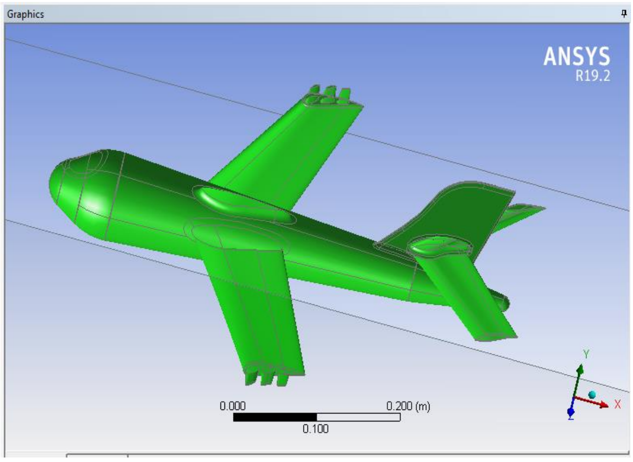

Fig 6.1 Existing aircraft construction.

6.2 MESHING OF EXISTING AIRCRAFT

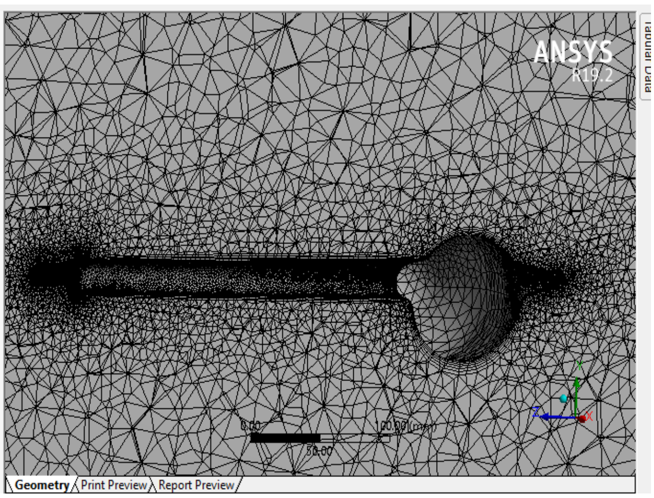

Fig 6.2 Meshing of existing model aircraft

6.3 DESIGN OF NEW MODEL AIRCRAFT WITH VERTEX GENERATOR

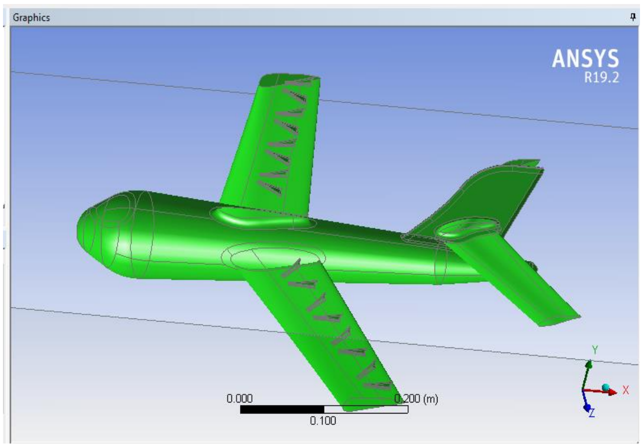

Fig 6.3 New model aircraft construction.

6.4 MESHING OF NEW MODEL AIRCRAFT GEOMENTRY

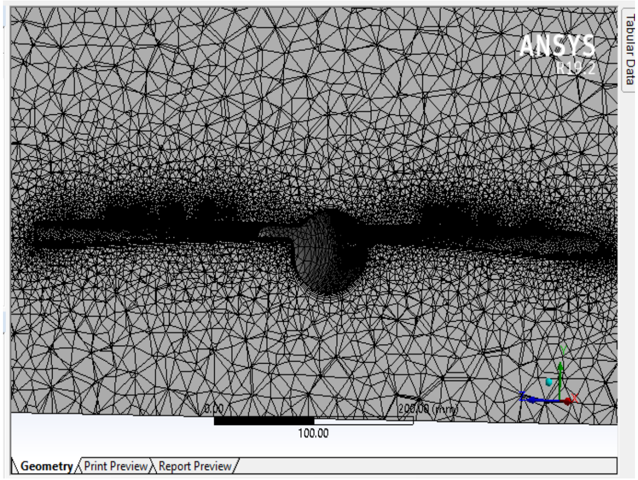

Fig 6.4 Meshing of new model aircraft

7. RESULT AND ANALYSIS

7.1 ANALYZING VELOCITY OF EXISTING AIRCRAFT

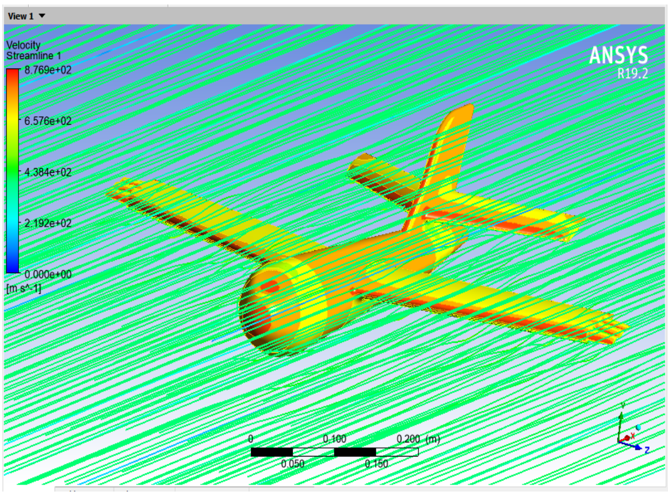

Fig 7.1 velocity diagram

7.2 ANALYZING PRESSURE OF EXISTING AIRCRAFT

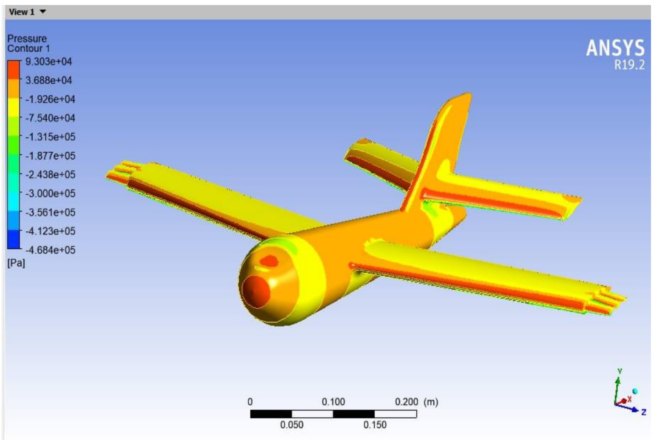

Fig 7.2 pressure diagram

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 03 | Mar 2019 www.irjet.net p-ISSN: 2395-0072

7.3 ANALYZING VELOCITY OF NEW MODEL AIRCRAFT

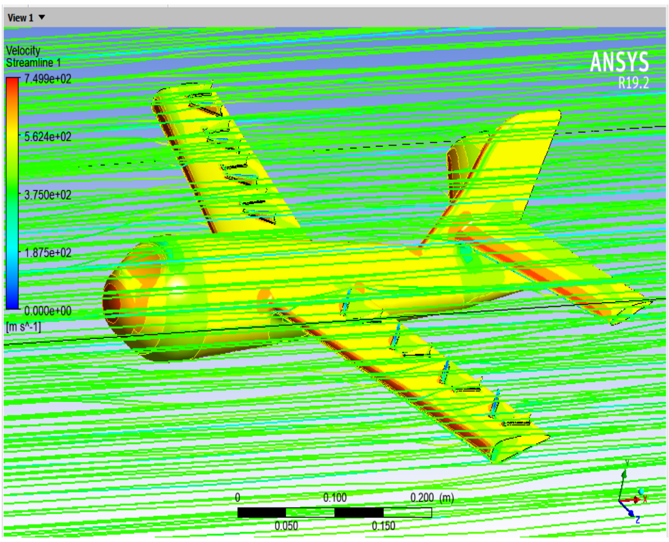

Fig 7.3 velocity diagram

7.4 ANALYZING PRESSURE OF NEW AIRCRAFT

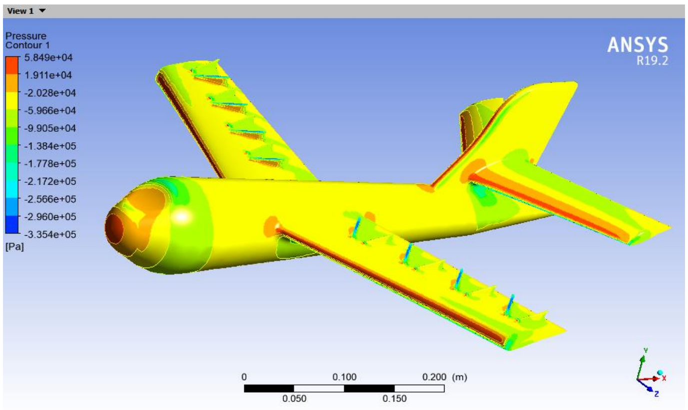

Fig 7.4 pressure diagram

7.5 CALCULATION OF DRAG FORCE FOR EXISTING MODEL

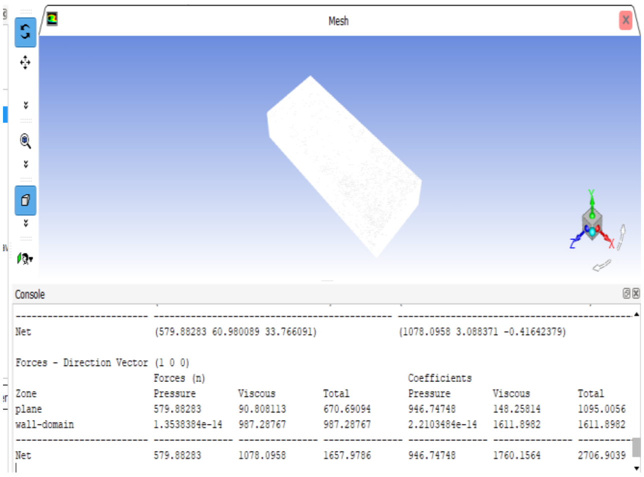

Fig 7.5 calculation of drag force

7.6 CALCULATION OF LIFT FORCE FOR EXISTING MODEL

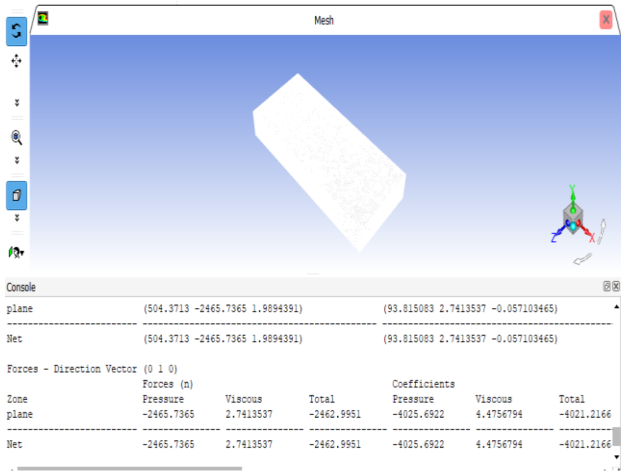

Fig 7.6 calculation of lift force

7.7 CALCULATION OF LIFT FORCE FOR NEW MODEL

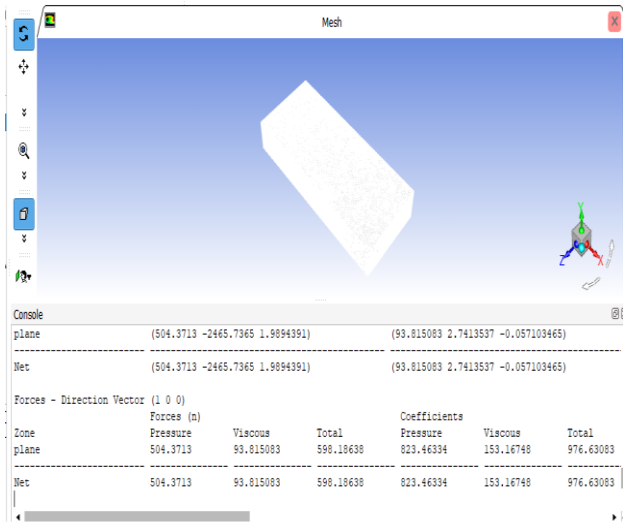

Fig 7.7 calculation of lift force

7.8 CALCULATION OF LIFT FORCE FOR NEW MODEL

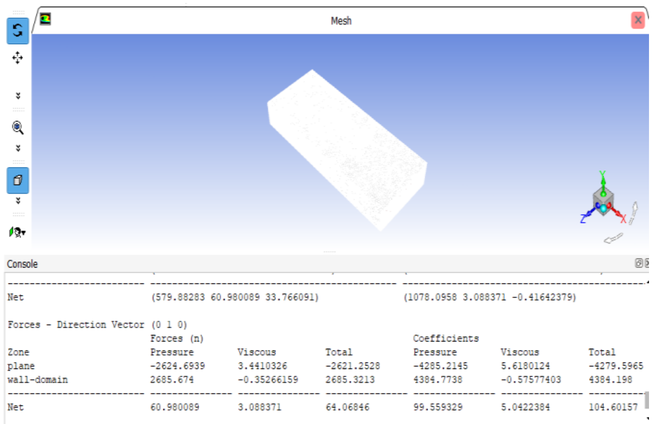

Fig 7.8 calculation of lift force

8. COMPARISON OF EXISTING AIRCRAFT VS NEW MODEL AIRCRAFT

8.1 TABULATION OF CALCULATED VALUES

S.No	PROPERTIES	EXISTING MODEL AIRCRAFT	NEW MODEL AIRCRAFT
1	LIFT FORCE (N)	-2465.73	-2624.69
2	DRAG FORCE (N)	579.88	504.37
3	PRESSURE (Pa)	9.303	5.849
4	VELOCITY (ms^-1)	8.769	7.499

NOTE: Negative (-) Sign indicates the lifting force from earth surface.

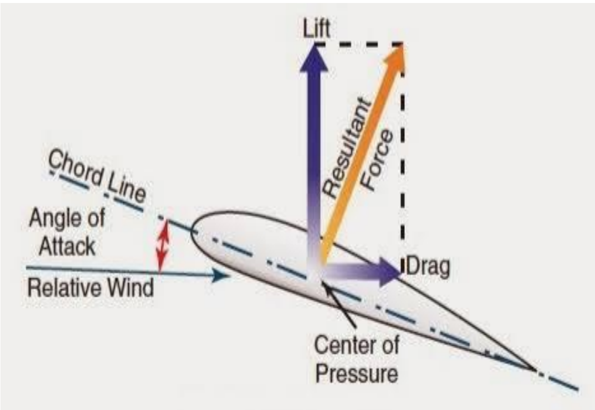

Fig 8.1 Drag force and lift force.

8.2 BAR CHART OF LIFT FORCE Vs DRAG FORCE

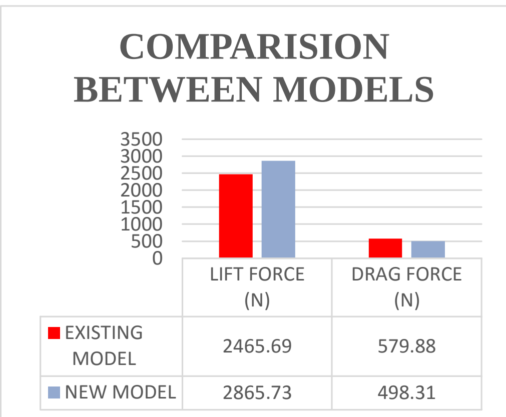

CHART-1: Bar chart of lift force Vs drag force

8.3 BAR CHART OF PRESSURE Vs VELOCITY

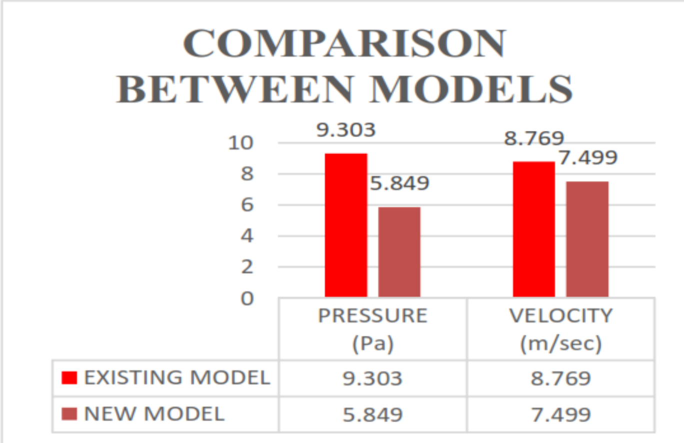

CHART-2: Bar chart of pressure Vs velocity

CONCLUSIONS

Drag forces and lift forces generated by aircraft is computed with low taper ratio of 0.125 by increasing angle of attack from 0°,5°,10°,15°,18°,20°. The lift and drag depend on the vertex generator dimensions, angle, shape& air foil shape on the winglet and it's also depending upon the velocity distribution and on the wing platform and on the wing area.

Hence, the drag force is decreased with increase in lift force with the help of vertex generator on both side of aircraft wings.

REFERENCES

1. CFD Analysis of Conceptual Aircraft Body, Manikantissar, Dr.Ankur geete, e-ISSN:2395-0056, p-ISSN:2395-0072, Volume 04,Issue: 01, Jan 2017.
2. Flow Analysis of Hal Tejas Aircraft Using Computational Fluid Dynamics with Different Angle of Attack, Abhishek Jain, S.K.Sharma, Abhishek Gour, ISSN: 2454-1311, Vol. 03, Issue:05, May 2017.
3. Computational Fluid Dynamics in Aerospace Industry in India, K.P.Singh, J.S.Mathur, V.Ashok, Vol. 60, No. 06, Nov-2010, pp. 639-652.
4. Flow Analysis Over an F-16 Aircraft Using Computational Fluid Dynamics, Manish Sharma, T.Ratna Reddy, Ch.Indira Priyadarsini, ISSN: 2250-2459, Vol. 3, Issue 05, May 2013.
5. CFD Simulations of Aircraft Body with Different Angle of Attack and Velocity, Shivasharanayya Hiremath, Anandkumar, S.Malipatil, ISSN: 2319-8753, vol. 3, Issue 10, October 2014.

6. Aerodynamic Analysis of Aircraft Wings Using CFD, Kimay Gore, Abhijeet Gote, Shailesh Humane e-ISSN:2395-0056, p-ISSN: 2395-0072, vol. 05, Issue: 06, June 2018.
7. Transisting Modelling for General CFD Applications in Aeronautics, R.B.Langtry, F.R.Menter, AIAA 2005-522.
8. CFD Analysis of Conceptual Aircraft Body, Manikantissar, Dr.Ankur geete, e-ISSN:2395-0056, p-ISSN:2395-0072, Volume 04,Issue: 01, Jan 2017.
9. Flow Analysis of Hal Tejas Aircraft Using Computational Fluid Dynamics with Different Angle of Attack, Abhishek Jain, S.K.Sharma, Abhishek Gour, ISSN: 2454-1311, Vol. 03, Issue:05, May 2017.
10. Computational Fluid Dynamics in Aerospace Industry in India, K.P.Singh, J.S.Mathur, V.Ashok, Vol. 60, No. 06, Nov-2010, pp. 639-652.
11. Flow Analysis Over an F-16 Aircraft Using Computational Fluid Dynamics, Manish Sharma, T.Ratna Reddy, Ch.Indira Priyadarsini, ISSN: 2250-2459, Vol. 3, Issue 05, May 2013.

BIOGRAPHIES

Mr.M.ALEXANDER
He was born in Dharmapuri, India in 1998.He completed his HSC with first class division in 2014-2015 and pursuing B.E Degree in Mechanical Engineering from KSR Institute For Engineering And Technology, Tamilnadu, India, in 2015-2019.

Mr.M.BALAJI
He was born in Mettur, India in 1997.He completed his HSC with first class division in 2014-2015 and pursuing B.E Degree in Mechanical Engineering from KSR Institute For Engineering And Technology, Tamilnadu, India, in 2015-2019.

Mr.K.N.BHARATHI
He was born in Karur, India in 1997.He completed his HSC with first class division in 2014-2015 and pursuing B.E Degree in Mechanical Engineering from KSR Institute For Engineering And Technology, Tamilnadu, India, in 2015-2019.

Mr.R.M.ELANTHENDRAL
He was born in HARUR, India in 1997.He completed his HSC with first class division in 2014-2015 and pursuing B.E Degree in Mechanical Engineering from KSR Institute For Engineering And Technology, Tamilnadu, India, in 2015-2019.

Mr.D.RAMKUMAR
He was born in Kallakurichi , India in 1991.He completed his Master Degree with first class division in 2015-2016 & Presently working as an Assistant Professor in KSR Institute For Engineering And Technology, Tamilnadu, India.