CFD Analysis of conceptual Aircraft body

Manikantissar [1], Dr.Ankur geete [2]

1 M. Tech scholar in Mechanical Engineering, SD Bansal college of technology, Indore, M.P, India

2 Associate professor in Mechanical Engineering Department, SD Bansal college of technology, Indore, M.P, India

---

ABSTRACT

The delta wing is a wing platform in the form of a triangle. Delta wings are most commonly seen on military aircraft. Delta wings are ideal for supersonic flight, and provide high maneuverability. In spite of its potential application in military aircraft, the understanding of the aerodynamics of such wings is far from complete. 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, 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 CFX.

Keywords: lift and drag force, conceptual aircraft model, CFD analysis, Angle of attack.

INTRODUCTION

From the development of the first powered flights (1903) to the present time, the study of the aerodynamic design has played an important role in the airplanes optimization. Traditionally it has been in the hands of the designer's experience, tests of flight and wind tunnel experiments, being this last tool the one that has provided a method of systematic study and the capability of making inexpensive adjustments of control parameters in a design. At the present time, Computational Fluid Dynamics (CFD) has come to complement the experimental studying, reducing the cost in tests and time for the generation of prototypes.

The selection of right wing is the most important aspect of airplane design which determines lift force generation, maneuverability stall angle, fuel storage. Delta wings finds its application for flying at supersonic speed and hence used for fighter aircraft and space shuttles. Delta wings also provide benefits of swept wings (decreased drag at supersonic speeds) due to their high sweep, and they are structurally efficient and provide a large internal volume which can be used for fuel tanks. They are also relatively simple and inexpensive to manufacture. At this point in the design process CFD analysis plays a crucial, if not its most important, role. Wind tunnel models are generally very expensive to build, costing perhaps hundreds of thousands of dollars or more, and wind tunnel test time is a significant cost driver during a project. In this work Analysis of aircraft body has done on CFD.

INTRODUCTION OF CFD ANALYSIS

CFD is a simulation tool used to predict what will happen, quantitatively when fluids flow, often with the complication of simultaneous flow of heat, mass transfer, phase change (melting, freezing, boiling), chemical reaction (combustion, rusting), mechanical movement of (pistons, fans etc.), stresses in and displacement of immersed or surrounding solids. CFD uses a computer to solve the relevant science-based mathematical equation, using information about the circumstances in question. There are three laws that have to be satisfied for the control volume, the conservation laws.

- **Conservation of mass**

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0$$

Conservation of momentum

$$\frac{\partial (\rho \mathbf{v})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \otimes \mathbf{v}) = -\nabla p + \nabla \cdot \tau + \rho \mathbf{g}$$

Conservation of energy

$$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho E \mathbf{v}) = \nabla \cdot (\kappa \nabla T) + \rho q - \nabla \cdot (p \mathbf{v}) + \mathbf{v} \cdot (\nabla \cdot \tau) + \nabla \mathbf{v} : \tau + \rho \mathbf{g} \cdot \mathbf{v}$$

$$E = e + \frac{|\mathbf{v}|^2}{2}, \qquad \frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho e \mathbf{v}) = \nabla \cdot (\kappa \nabla T) + \rho q - p \nabla \cdot \mathbf{v} + \nabla \mathbf{v} : \tau$$

Literature review

Karna S. Patel et al. (1) studied the CFD analysis of the flow over NACA 0012 airfoil and conclude that at the zero degree of AOA there is no lift force generated. The amount of drag force and value of drag coefficient also increased but the amount of increment in drag force and drag coefficient is quite lower compare to lift force.

In this study, NACA 2412 airfoil [2] is used to design the wing, in which the first digit is the maximum camber in hundredths of the chord, the second digit is the location of the maximum chamber from the leading edge in tenths of the chord, and the last two digits represent the maximum thickness in hundredths of the chord [3]. The parameters are chosen, such as airfoil chord c = 0.3m, airfoil span l = 1.6m . These dimensions are used to fabricate the experimental wing model, which are also consistent with the open data of a number of test UAVs samples in Vietnam.

Zoran A. Stefanović has used CFD technique to analyse light aircraft at different stages .This paper describes the calculation tools and methods applied during the aerodynamic analyses of a new light aircraft at different development stages, and compares the results obtained by them, with the aim to verify and support the above statement, considering light aircraft aerodynamic design. Comparison of lift forces/drag forces has been made at for angle of attack at 0 0 and 300[4].

S. Rajiv Rao[5] has performed CFD analysis on VTOL aircraft i.e. vertically take off and landing type of aircraft. Model taken is V-22 osprey tilt rotor aircraft designed to operate under a wide range of flight conditions. Studies has been made at different angle of attack from 0 – 20 degrees and coefficient of lift and coefficient of drag is calculated for each angle of attack .CL & Cd values increase with angle of attack up to 6 0 beyond which these coefficients change abruptly. [6]Abhishek Patil has computed the 3D turbulent flow field over sharp edged finite wings with a diamond shaped planforms and moderate sweep angle . Drag force and lift force were analyzed at different angle of attack 0,4,8,12 degrees . relative performances of different sections are assessed by comparing their surface pressuredistribution, skin friction distribution, flow pattern and variation of aerodynamic coefficients

WING PARAMETER

In design and analysis of wing, the wing parameters play an important role. The wing parameter gives the actual operations about the wing analysis. Some wing parameters are-

- Wing reference (or plan form) area (Sref or S)
- Number of the wings
- Vertical position relative to the fuselage (high, mid, or low wing)
- Horizontal position relative to the fuselage
- Cross section (or airfoil)
- Aspect ratio (AR) Taper ratio
- Tip chord (Ct)
- Root chord (Cr)
- Mean Aerodynamic Chord (MAC or C)
- Span (b)
- Twist angle (or washout)
- Sweep angle
- Dihedral angle
- Incidence etc.

DEVELOPING CAD MODEL

CAD model is first prepared in CREO 2.0 and is converted in .iges format to export it to ANSYS CFXCreo is developed by parametric technology corporation and this is one of the fastest growing solid modeling software. As a parametric featured based solid modeling tool, it not only unites the three dimensional model (3D) parametric features with 2D tools, but also addresses every design-through-manufacturing process. The solid modeling tool used here allows us to easily import the standard format files with an amazing compatibility to other software's

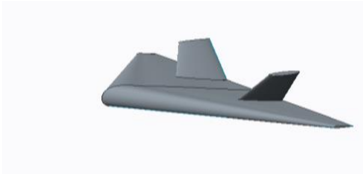

Fig 1.0 CAD model of conceptual aircraft

ANSYS SIMULATION

The CAD model is imported in ANSYS CFX and enclosure is created . Using boolean operation continuum is generated The appropriate size and shape of the computational domain, also referred to as control volume, and the best placement of the model in the domain, needs to be determined. A domain too large will make the simulation unnecessarily large and waste computational resources, however a domain too small will lower the accuracy of the results. The properties of the domain such as temperature, pressure and fluid properties need to be chosen.

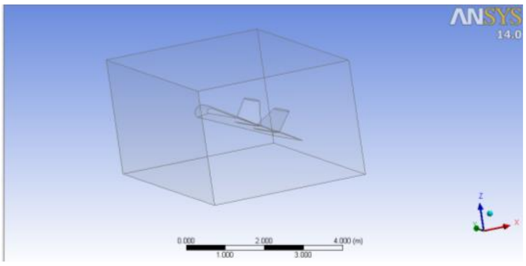

Fig 2.0 conceptual aircraft domain

Boundary Conditions: The conditions at the boundary of the domain need to be set such as inlet velocities, outlets and wall attributes.

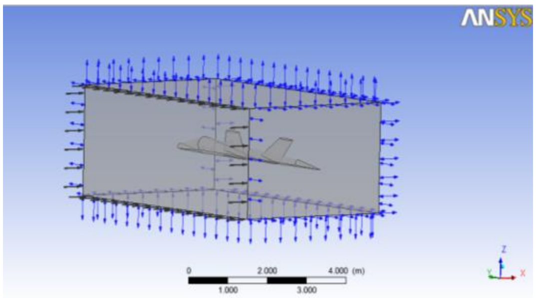

Fig 3.0 Boundary conditions aircraft domain

Discretization of the Domain: Since CFD utilizes numerical solutions the domain needs to be discretezised or meshed as it is more commonly referred to. The mesh will have to be refined in areas with high gradients for example close to the surface around the aircraft model. Initial Values Initial values need to be set for all the nodes in the domain.

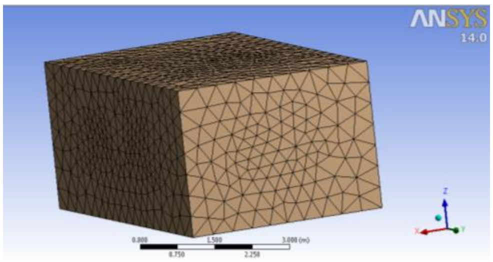

Fig 4.0 Mesh model aircraft

Post Processing : The simulation results will give the sums of the forces acting in each direction on the model, or any chosen part of it. Through this, the drag coefficient CD can be obtained. The amount of lift created by the wing will have to be taken into account in order to be able to calculate the induced drag. Subtracting the induced drag from the total drag should give the zero lift drag.

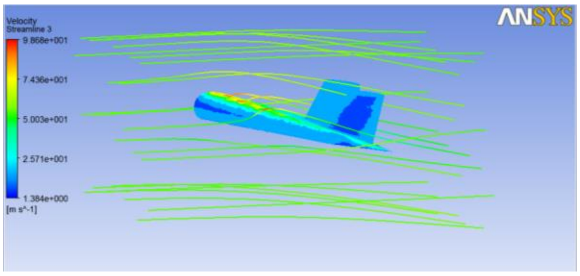

Fig 5.0 Simulation model of aircraft

METHODOLOGY:-

In all of these approaches the same basic procedure is followed. During pre-processing the geometry (physical bounds) of the problem is defined. The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform. The physical modeling is defined – for example, the equations of motion + enthalpy + radiation + species conservation. Boundary conditions are defined. This involves specifying the fluid behavior and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined. The simulation is started and the equations are solved iteratively as a steady-state or transient. Finally a postprocessor is used for the analysis and visualization of the resulting solution.

Wing Design Parameters

Aircraft design involves various design parameters. For the present study will focus our study on delta wing design of our aircraft and at the same time keeping other airplane body parameters constant. The design parameters involved are:

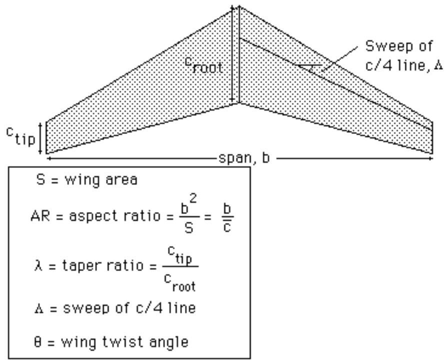

Fig 6.0 Design details of Aircraft wing

1. Wing area (SW or Sref or S)

$$S = \int_{-b/2}^{b/2} c(y)\,dy$$

2. Wing span b , is measured from tip to tip.
3. Wing average chord , is the geometric average . The product of span and average chord is the wing area

$$\bar{c} = \frac{1}{b}\int_{-b/2}^{b/2} c(y)\,dy = \frac{S}{b}$$

4. Aspect ratio is defined as

$$AR = \frac{b^2}{S}$$

5. Root chord is the chord of wing centerline and tip chord is the chord of the tip
6. Taper ratio

$$\lambda = \frac{c_t}{c_r}$$

7. Mean aerodynamic chord

$$\tilde{c} = \frac{1}{S}\int_{-b/2}^{b/2} [c(y)]^2\,dy$$

RESULTS AND DISCUSSION

Analysis is performed for different taper ratios and pressure lift force and drag force is computed . CL vs angle of attack and CD vs angle of attack is computed

Taper ratio is given by

$$\lambda = \frac{c_t}{c_r}$$

Inlet velocity : 55 m/s
Turbulence Model: K-Epsilon

$$\rho\frac{\partial k}{\partial t} + \rho\vec{u}\cdot\nabla k = \nabla\cdot\left[\left(\mu + \frac{\mu_T}{\sigma_k}\right)\nabla k\right] + \mu_T \mathrm{P}(\vec{u}) - \rho\varepsilon$$

CASE 1: Taper ratio is .125
Angle of attack is 0⁰

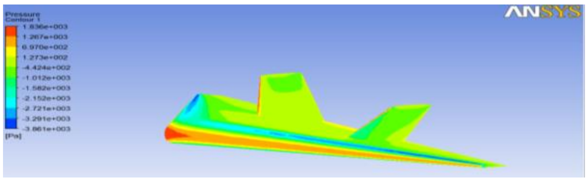

Fig 7.0 Pressure Contour of Aircraft

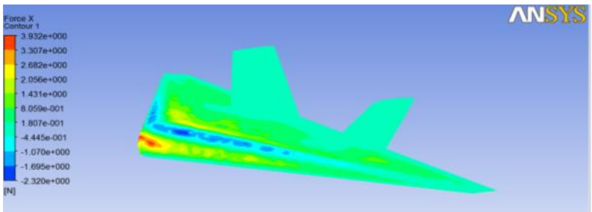

Fig 8.0 Drag Force of Aircraft

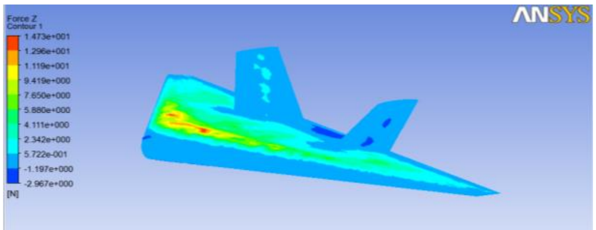

Fig 9.0 Lift Force of Aircraft

Angle of attack is 5⁰

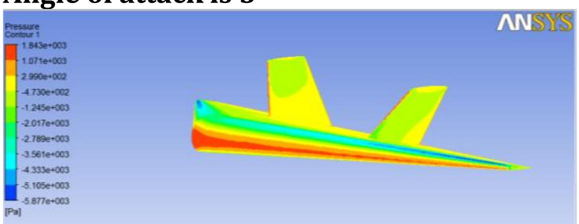

Fig 10.0 Pressure Contour of Aircraft

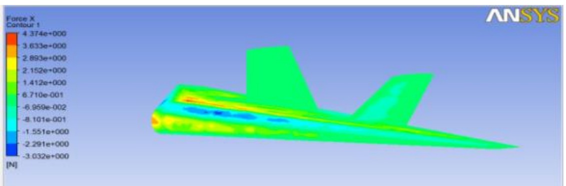

Fig 11.0 Drag Force of Aircraft

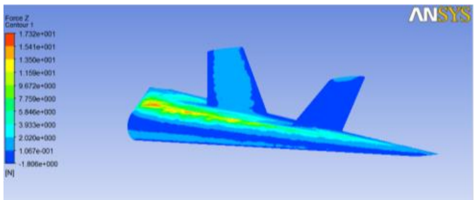

Fig 12.0 Lift Force of Aircraft

CASE 2: Taper ratio is .4
Angle of attack is 0⁰

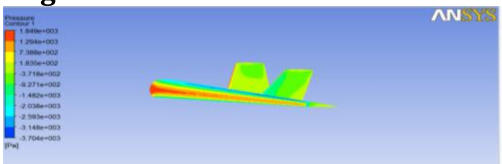

Fig 13.0 Pressure Contour of Aircraft

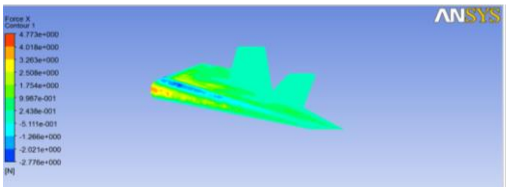

Fig 14.0 Drag force of Aircraft

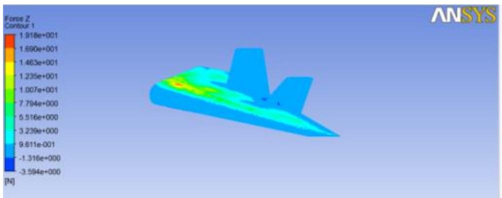

Fig 15.0 Lift Force of Aircraft

Angle of attack is 5⁰

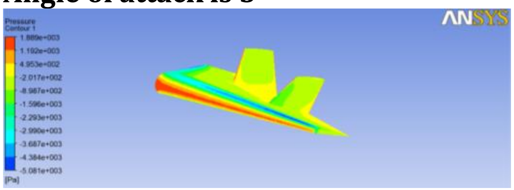

Fig 16.0 Pressure Contour of Aircraft

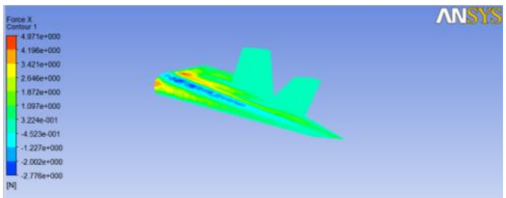

Fig 17.0 Drag force of Aircraft

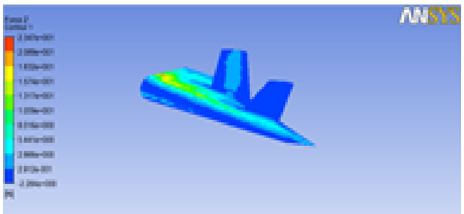

Fig 18.0 Lift force of Aircraft

CASE 1: Taper ratio= 0.25
Table no.1 Calculated values

Angle of Attack	Pressure (Pa)	Drag Force (N)	Lift Force (N)	C_L	C_D
0^0	1836	3.932	14.73	.0006354	.00016
5^0	1843	4.37	17.32	.0007471	.00018
10^0	1821	6.73	19.93	.0008597	.00029
15^0	1876	10.47	23.12	.0009974	.00041
18^0	1947	12.73	26.89	.001160	.00054
20^0	1938	14.35	26.29	.0011303	.00061

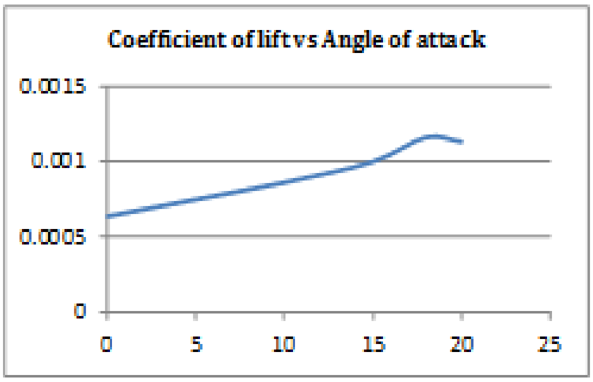

Fig 19.0 Coefficient of lift vs Angle of attack graph

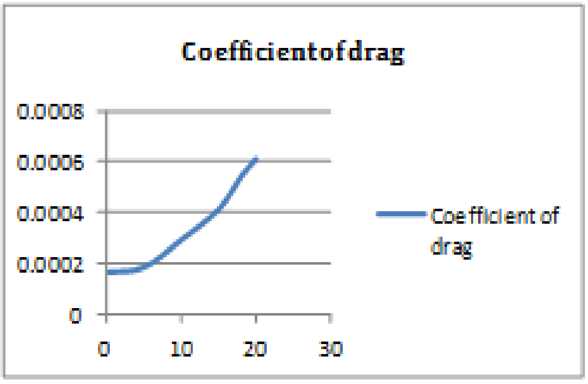

Fig 20.0 Coefficient of Drag vs Angle of attack graph

CASE 2: Taper ratio=0.4
Wing Area:15.47 m²
Table no.2 Calculated values for Case 2.0

Angle of Attack	Pressure	Drag Force	Lift Force	C_L	C_D
0^0	1849	4.77	19.18	.000660	.000164
5^0	1889	4.97	23.47	.000808	.000171
10^0	1896	6.94	20.31	.000699	.000239
15^0	1919	10.19	28.75	.000990	.000351
18^0	1966	12.90	26.56	.000915	.000444
20^0	2071	15.89	31.17	.001074	.000547

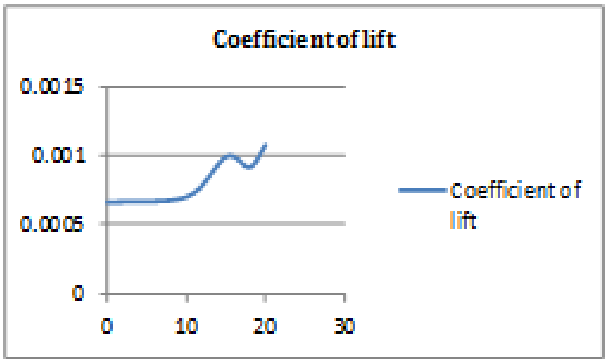

Fig 21.0 Coefficient of lift vs Angle of attack graph

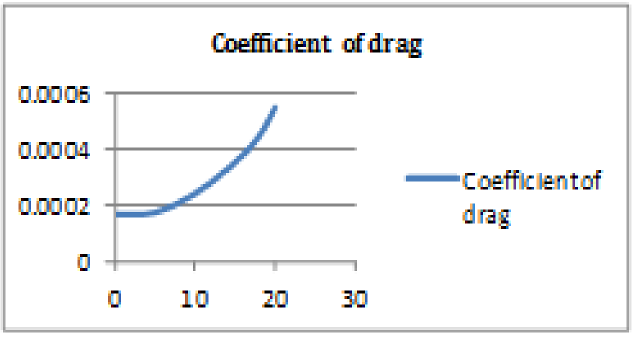

Fig 22.0 Coefficient of drag vs Angle of attack graph

CONCLUSION:-

The computation of flow past a biconvex delta wing cross section with aspect ratio shows physically possible flow field, demonstrated by the surface pressure distribution and particle traces on different planes.Drag force and lift force generated by aircraft is computed with low taper ratio of .125 by increasing angle of attack from 0^0,5^0,10^0,15^0, 18^0,20^0.Drag force and lift force generated by aircraft is computed with high taper ratio of .4 by increasing angle of attack from 0^0,5^0,10^0,15^0, 18^0,20^0.

Coefficient of lift vs angle of attack , coefficient of drag vs angle of attack graph is plotted .Stall angle with low taper ratio is found to be at 18^0 angle of attack and stall angle with high taper ratio if found to be at 15^0 angle of attack.Formation of vortex has been noticed at back portion of aircraft is shown by vectorial representation.The lift and drag depend on the airfoil shape and it is depending upon the velocity distribution, but also on the wing planform and on the wing area. It is possible to calculate the aerodynamic properties of differently sized airfoils or wings if all forces and moments are normalized Turbulence kinetic energy is found to be maximum at front portion of wing with value of 32.65 m^2s^{-2}

REFERENCES

[1] Karna S. Patel, Saumil B. Patel,Utsav B. Patel, Prof. Ankit P. Ahuja "CFD Analysis of an Aerofoil" International Journal of Engineering Research Volume No.3, Issue No.3, pp : 154-158, ISSN:2319-6890)(online),2347-5013(print) 01 March 2014

[2] Prabhakar A. and Ohri A., "CFD Analysis on MAV NACA 2412 Wing in High Lift Take-Off Configuration for Enhanced Lift Generation", J Aeronaut Aerospace Eng. 2: 125. doi:10.4172/2168-9792.1000125, 2013.

[3] Ira H. Abbott and Albert E. Von Doenhoff, "Theory of Wing Sections", Dover Publishing, New York, 1951.

[4] FME Transactions (2014) 42, VOL. 42, No 2, 2014 Ǝ 95

[5] International Journal of Engineering Research & Technology (IJERT) IJERT ISSN: 2278-0181 IJERTV3IS090267 www.ijert.org Vol. 3 Issue 9, September- 2014

[6] INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICALENGINEERING , ISSN (ONLINE): 2321-3051 Vol.2 Issue 3.