International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 4, Issue 11
November 2017

Computational Aerodynamic Analysis of F-16 Falcon Wing

Babitha Kodavanla[1], Dr P K Dash[2] , Dr P Srinivas Rao[3],
[1,]Assistant Professor,[2,3]Professor
[1,2,3]Departmentof Aeronautical Engineering,
Institute of Aeronautical Engineering, Dundigal, Hyderabad, India.

ABSTRACT:

In order to reduce the risk to aircrews during testing and the costs incurred by extensive wind tunnel and flight tests, it would be helpful to have a CFD tool which enabled engineers and designers to analyze and evaluate optimized designs. The main objective of project is to evaluate the aerodynamics characteristics of high performance behavior of the Lockheed Martin F-16 Fighting Falcon wing from low to high angles of attack by using different Numeric technical approaches. This project outlines the development of a computational model of the F-16 Fighting Falcon wing model in a finite computational domain, segmentation of this domain into discrete intervals, application of the boundary condition such as Mach number and angles thus obtaining the plots and results for coefficient of pressure, lift coefficient, drag coefficient, etc. The CFD simulations have been done in subsonic, transonic and supersonic flight regimes. Finally obtaining the optimized results of lift and drag forces.

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 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 find 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 role The design of F-16 falcon which takes full advantage of the unique capabilities of the CFD environment while minimizing the computational time on increasing the high-performance characteristics using different types of technical approaches **[1-2].** The geometric significance of the F-16 Falcon aircraft and its behavior at different angle of attacks **[3]**is studied. Hence the geometric modeling of F-16 Falcon is designed using Catia with desired parameters **[3].** The meshing, which plays a significant role for the analysis of F-16 and representative results will be given by each different type of computational numerical approaches [**4].** Both structured and unstructured grid types were used for Computational analysis. Finally obtaining the accurate values for grid refinements ensuring the solutions for the different flight conditions, subsonic, transonic and supersonic conditions from low to high angles of attack. Hence, the importance of CFD tools which plays an significant role in minimization of time, Cost and also optimized design phase is obtained **[5].** CFD predictions over laminar and turbulent cases are studied during result analysis. The CFD solutions used in the study are for finding forces and moments coefficients of the F-16 Falcon aircraft from different solvers subjecting to different Mach (M) at different angle of attack (α). The flow properties on the wing, Shock waves formed on the leading edge surface of the wing both at upper and lower surfaces with change in Mach at different angles can be seen. The results are validated with the peer data available **[8].** The results obtained for C_l, C_d, Cm and the moments on the Lockheed F-16 Falcon are accurate at the desired cases considered for different Mach(M) and Angle of attacks(α).

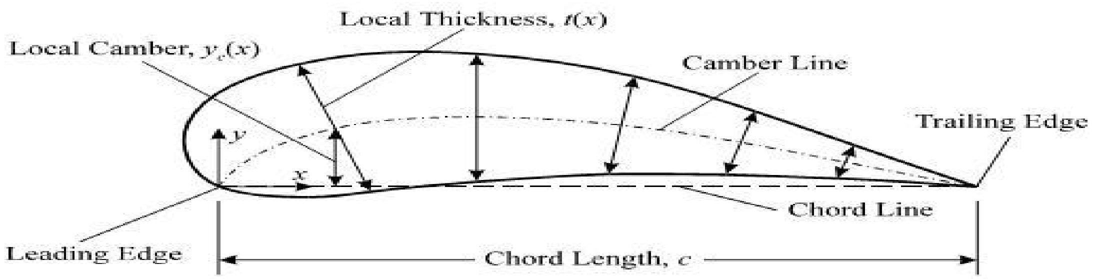
Design of Wing:

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. Wing reference area, Vertical position relative to the fuselage (high, mid, or low wing), Horizontal position relative to the fuselage, Cross sectional area, Aspect ratio (AR), Taper ratio, Tip chord(C_t),Root chord (C_r),Mean Aerodynamic Chord (MAC or C),Span (b),Twist angle(or washout) Sweep angle, Dihedral angle, Incidence etc. are the design parameters for a wing.

Airfoil Selection

The NACA have some special standards for every Airfoil design, as F16 have the code is 64A204 or 64A206. Due to this code, we can get the overall profiles of Airfoil. It is appropriate to claim that the Airfoil section is the second most important wing parameter; after wing plan form area. The Airfoil section is responsible for the generation of the optimum pressure distribution on the top and bottom surfaces of the wing such that the required lift is created with the lowest aerodynamic cost. Here the airfoil selected for the falcon is NACA6 series that is NACA-64-209 or NACA-64-208. These six series give us a different way to design the Airfoil. The Airfoil of F-16 which helps in increased fuselage blending of aircraft, at root, the increased thickness helps in more fuel capacity, which increases the range of the aircraft.



Design of wing

The delta wing specially designed for the fighter planes for having max Speed and taking cruise and loiters. The wing selected that is delta wing. In design, the basic configuration arrangement, size, shape, and lengths are calculated and modeled.

Calculated design parameters are as follows:

Wing Area=600sq ft.

Span=32ft 48inch.

Aspect Ratio=1.750.

Taper Ratio=0.100.

Sweep Back angle=40 Deg.

Root Airfoil=NACA-64-209, Tip Airfoil=NACA-64-208.

Meshing of the wing:

ICEM-CFD is used for grid generations which are very highly adoptable mesh. Two different types of grids were used for the meshing. One is structured and unstructured mesh. Surface triangulation of common unstructured and after local refinement the as mesh refinement is used for getting accurate results, the size and shape of the mesh is calculated with respect to AR of wing. Mesh checking is also an important process where in which for obtaining accurate results of the falcon wing.

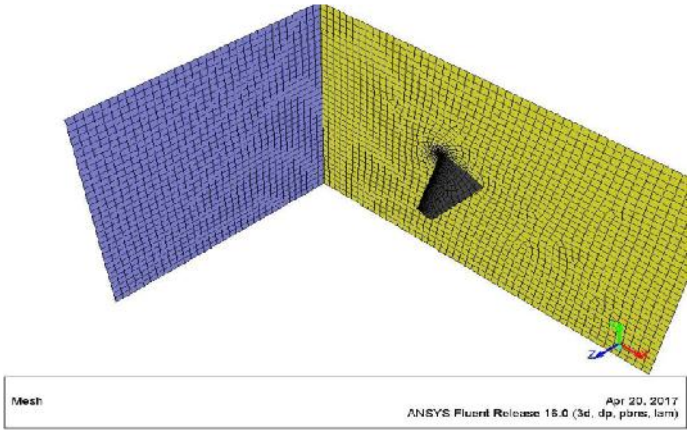


Fig.1 Meshing plot of F-16 wing

Method:

Navier-Stokes Equations
Conservation Law

Navier-Stokes equations are the governing equations of Computational Fluid Dynamics. It is based on the conservation law of physical properties of fluid. The principle of conservational law is the change of properties, for example mass, energy, and momentum, in an object is decided by the input and output.

For example, the change of mass in the object is as follows

$$\frac{dM}{dt} = \dot{m}_{in} - \dot{m}_{out} \qquad (1)$$

If $\dot{m}_{in} - \dot{m}_{out} = 0$, we have

$$\frac{dM}{dt} = 0 \qquad (2)$$

Which means

$$M = const \qquad (3)$$

Navier-Stokes Equation

Applying the mass, momentum and energy conservation, we can derive the continuity equation, momentum equation and energy equation as follows.

Continuity Equation

$$\frac{D\circ}{Dt} + \circ \frac{\partial U_i}{\partial x_i} = 0 \qquad (4)$$

Momentum Equation

$$\underbrace{\circ \frac{\partial U_j}{\partial t}}_{I} + \underbrace{\circ U_i \frac{\partial U_j}{\partial x_i}}_{II} = \underbrace{-\frac{\partial P}{\partial x_j}}_{III} \underbrace{-\frac{\partial \text{²}_{ij}}{\partial x_i}}_{IV} + \underbrace{\circ g_j}_{V} \qquad (5)$$

Where

$$\text{²}_{ij} = -\text{«}\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) + \frac{2}{3}\text{¢}_{ij}\text{«}\frac{\partial U_k}{\partial x_k} \qquad (6)$$

I: Local change with time
II: Momentum convection
III: Surface force
IV: Molecular-dependent momentum exchange (diffusion)
V: Mass force**Energy Equation**

$$\underbrace{\circ c_{\text{«}} \frac{\partial T}{\partial t}}_{I} + \underbrace{\circ c_{\text{«}} U_i \frac{\partial T}{\partial x_i}}_{II} = \underbrace{-P\frac{\partial U_i}{\partial x_i}}_{III} + \underbrace{\text{a}\frac{\partial^2 T}{\partial x_i^2}}_{IV} \underbrace{-\text{²}_{ij}\frac{\partial U_j}{\partial x_i}}_{V} \qquad (7)$$

I : Local energy change with time
II: Convective term
III: Pressure work
IV: Heat flux (diffusion)
V: Irreversible transfer of mechanical energy into heat
If the fluid id compressible, we can simplify the continuity equation and momentum equation as follows.

Continuity Equation

$$\frac{\partial U_i}{\partial x_i} = 0 \qquad (8)$$

Momentum Equation

$$\circ \frac{\partial U_j}{\partial t} + \circ U_i \frac{\partial U_j}{\partial x_i} = -\frac{\partial P}{\partial x_j} - \text{«}\frac{\partial^2 U_j}{\partial x_i^2} + \circ g_j \qquad (9)$$

General Form of Navier-Stokes Equation

To simplify the Navier-Stokes equations, we can rewrite them as the general form.

$$\frac{\partial(\circ\Phi)}{\partial t} + \frac{\partial}{\partial x_i}\left(\circ U_i \Phi - \Gamma_\Phi \frac{\partial \Phi}{\partial x_i}\right) = q_\Phi \qquad (10)$$

When $\Phi = 1, U_j, T$, we can respectively get continuity equation, momentum equation and energy equation.

Finite Volume Method

The Navier-Stokes equations are analytical equations. Human can understand and solve them, but if we want to solve them by computer, we have to transfer them into discretized form. This process is discretization. The

ISSN 2394 – 3386
Volume 4, Issue 11
November 2017

typical discretization methods are finite difference, finite element and finite volume methods. Here we introduce finite volume method.

The Approach of Finite Volume Method

Integrate the general form of Navier-Stokes equation over a control volume and apply Gauss Theory

$$\int_V \frac{\partial}{\partial x_i} \Phi dV = \int_S \Phi \cdot n_i dS \qquad (11)$$

We can get the integral form of Navier-Stoke equation

$$\int_V \frac{\partial(\circ\Phi)}{\partial t} dV + \int_S \left(\circ U_i \Phi - \Gamma \frac{\partial \Phi}{\partial x_i} \right) \cdot n_i dS = \int_V q_\Phi dV \qquad (12)$$

To approximate the volume integral, we can multiply the volume and the value at the center of the control volume. For example, we have a 2D domain as fig 2. To approximate the mass and momentum of control volume P, we have

$$m = \int_{V_i} \circ \; dV \approx \circ_p V \,, \quad mu = \int_{V_i} \circ_i u_i dV \approx \circ_P u_P V \qquad (13)$$

To approximate the surface integral, for example pressure force, we have

$$\oint_{S_i} P \, dS \approx \sum_k P_k \; S_k \qquad k = n, s, e, w \qquad (14)$$

Normally we store our variables at the center of control volume, so we need to interpolate them to get P_k, which are located at the surface of control volume.

Typically, we have two types of interpolations, one is upwind interpolation, and the other one is central interpolation.

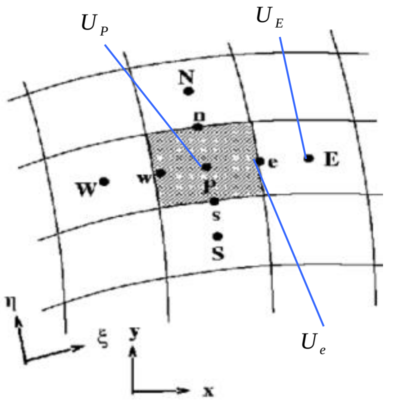


Figure 2 2D Structured Grid Domains

Upwind Interpolation

$$U_e = \begin{cases} U_P & \text{if } \; (\vec{U} \cdot \vec{n})_e > 0 \\ U_E & \text{if } \; (\vec{U} \cdot \vec{n})_e < 0 \end{cases}$$

Central Interpolation

$$U_e = U_E \, \text{a}_e + U_P (1 - \text{a}_e) \qquad \text{a}_e = \frac{x_e - x_P}{x_E - x_P}$$

International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 4, Issue 11
November 2017

Conservation of Finite Volume Method

If we use finite difference and finite element approach to discretized Navier-Stokes equation, we have to manually control the conservation of mass, momentum and energy. But with finite volume method, we can easily find out that, if the Navier-Stokes equation is satisfied in every control volume, it will automatically be satisfied for the whole domain. In another words, if the conservation is satisfied in every control volume, it will be automatically satisfied in whole domain. That is the reason why finite volume is preferred in computational fluid dynamics.

Methodology:

.The CFD simulation of continuum was done in ANSYS Fluent 16. CFD capability, the for excellent opportunity to investigate the use of new technologies to help solve complex problems. Capabilities such as grid adaptation, new turbulence models, and new numerical algorithms were all investigated. These studies provided a great deal of information that will help with the development and deployment of these new technologies. In this initially the meshing of the continuum was read and then checked. The turbulence models used for these simulations were the Inviscid, and Laminar. The geometry used and the mesh generated was the same for all cases, including the transonic Condition. The solver was based on “Pressure based” as the all of the simulations were compressible. Navier Stokes Energy equations were selected in order to solve the cases as all of the simulations were compressible. The working fluid in this simulation was ideal gas as the boundary condition “Pressure Far field” was compactable with it. It was considered theF-16 aircraft was flying at sea level conditions and the viscosity was solved using Sutherland equations. Pressure far-field boundary conditions were used in this simulation to model a free-stream compressible Flow at infinity, with free-stream Mach number and static conditions specified. The wing was given with “wall” boundary condition. The solver was semi-implicit method for pressure-linked equations (SIMPLE) algorithm. This algorithm is an iterative procedure for solving equations for velocity and pressure, for steady-state. under-relaxation factors for momentum=0.7andpressure=0.3. For the discretization, the pressure was kept as standard, while the other parameters as Second Order Upwind. Monitoring the convergence during the solution was dynamically checked by force coefficient values rather than checking for the convergence through residuals. The data were printed, reported and displayed in plots of lift, drag, and moment coefficients, and residuals for the solution variables. In the Force Monitors, the Force vectors Lift and Drag had to define with relative to the free stream direction.

Results:

AOA (Angle of attack)	Mach(M)	Lift Coefficient		Drag Coefficient			Moment	
		$C_{L\,Max}$	$C_{L\,Min}$	$C_{D\,Max}$	$C_{D\,Min}$	C_{L}/C_{D}	$C_{m\,Max}$	$C_{m\,Min}$
0^0	0.6	4.5	3.8	0.22	1.7	20.45	0.36	0.38
	1	4.0	0.5	6.7	8.7	0.59	0.12	0.058
	1.2	1.5	-0.1	10	12	0.15	0.21	0.13
	1.5	-3.0	-9.0	15	17.5	-0.2	0.34	0.25

Table 1. Flight conditions F-16, force and moment coefficients at 0 degree AOA.

AOA	Mach(M)	Lift(L)	Drag(D)		Moment

(Angle of attack)		$C_{L\ Max}$	$C_{L\ Min}$	$C_{D\ Max}$	$C_{D\ Min}$	C_L/C_D	$C_{m\ Max}$	$C_{m\ Min}$
5^0	0.6	22.5	21.0	1.3	2.9	17.30	-0.38	-0.42
	1	64	54	11.5	12.3	5.56	-0.78	-0.91
	1.2	62.5	57	14.25	16.25	4.38	-0.78	-0.90
	1.5	82.5	70	21.0	24.8	3.92	-1.0	-1.2

Table 2. Flight conditions F-16, force and moment coefficients at 5 degree AOA.

AOA (Angle of attack)	Mach(M)	Lift(L)		Drag(D)			Moment	
		$C_{L\ Max}$	$C_{L\ Min}$	$C_{D\ Max}$	$C_{D\ Min}$	C_L/C_D	$C_{m\ Max}$	$C_{m\ Min}$
10^0	0.6	40	36	4.25	7.0	9.41	-0.68	-0.78
	1	115	110	25	26	4.6	-1.58	-1.8
	1.2	140	120	32	34	4.37	-1.76	-2.24
	1.5	172	150	42	46	4.09	-2.25	-2.6

Table 3. Flight conditions F-16, force and moment coefficients at 10 degree AOA.

AOA (Angle of attack)	Mach(M)	Lift(L)		Drag(D)			Moment	
		$C_{L\ Max}$	$C_{L\ Min}$	$C_{D\ Max}$	$C_{D\ Min}$	C_L/C_D	$C_{m\ Max}$	$C_{m\ Min}$
15^0	0.6	52	40	9.0	13.5	5.7	-0.74	-1.0
	1	165	150	50	55	3.3	-2.4	-2.6
	1.2	200	175	58	60	3.44	-3.0	-3.2
	1.5	270	225	65	84	4.15	-3.5	-4.2

Table 4. Flight conditions F-16, force and moment coefficients at1 5 degree AOA.

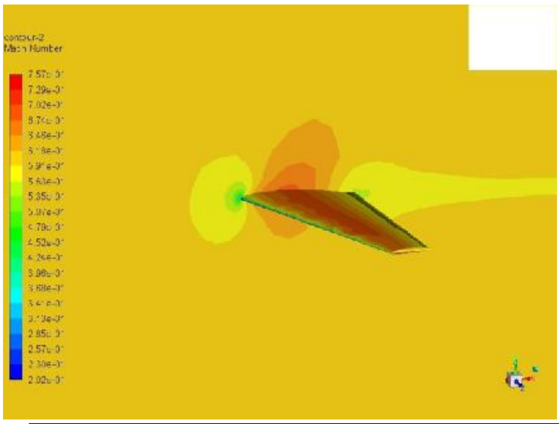


Fig.2 Mach Counters at M=0.6at 0^0AOA

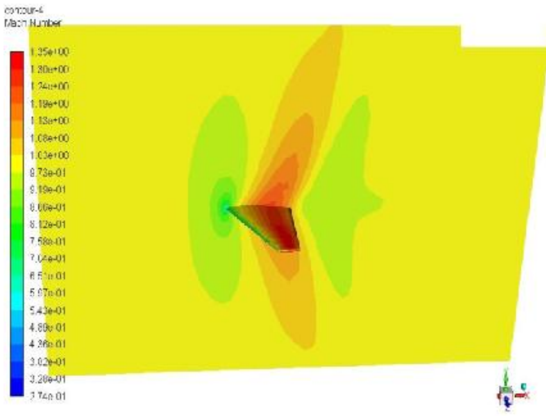


Fig.3 Mach Counters at =1 at 5^0AOA

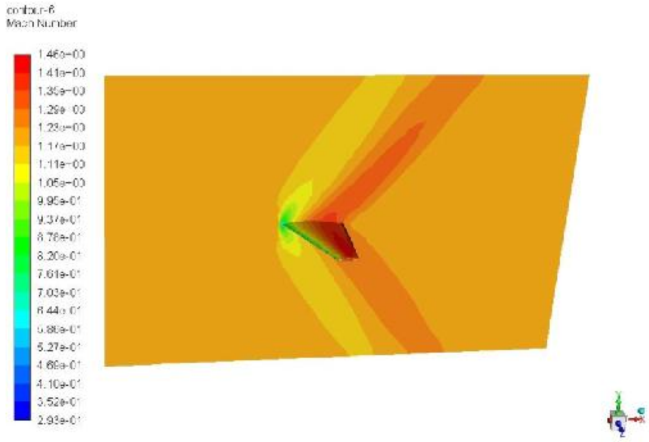


Fig.4 Mach Counters at M=1.2at 10^{0}AOA

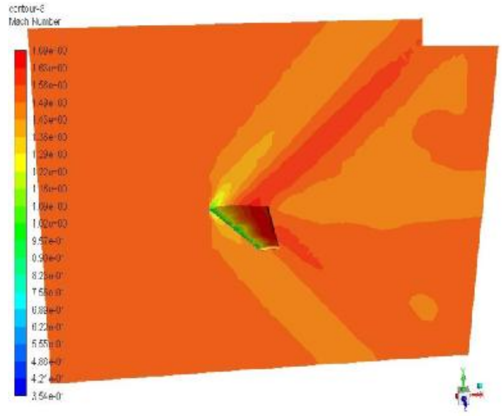


Fig.5 Mach Counters at M=1at 10^{0}AOA

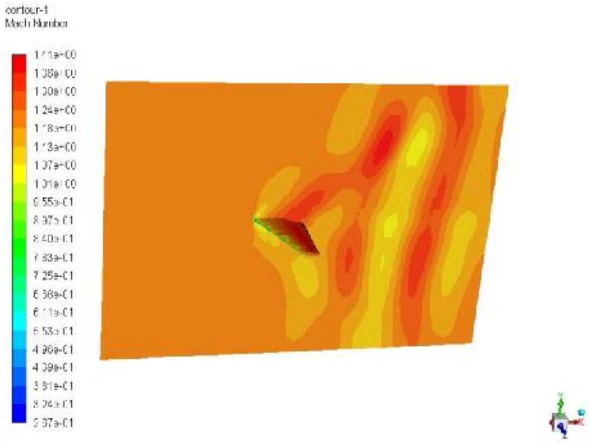


Fig.6 Mach Counters at M=1.2at 15^{0}AOA

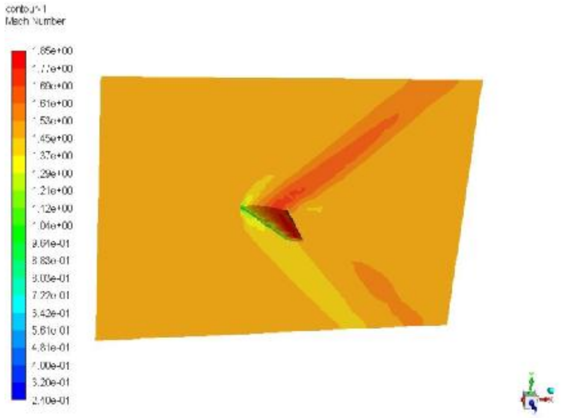


Fig.7 Mach Counters at M=1.5at 15^{0}AOA

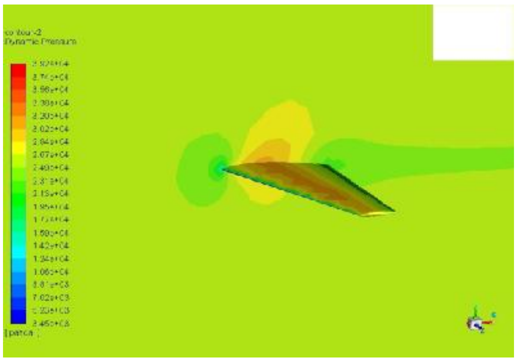


Fig.8 Dynamic pressure at M=0.6 at 0^{0}AOA

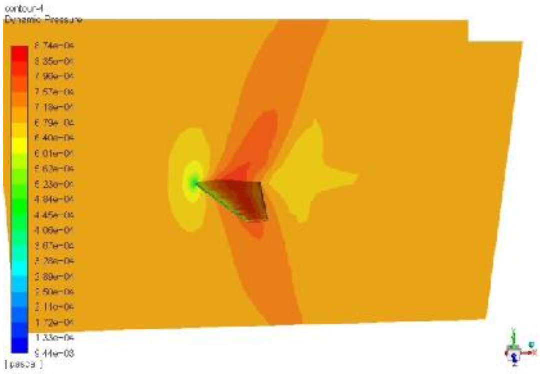


Fig.9 Dynamic pressure at M=1at 0^{0}AOA

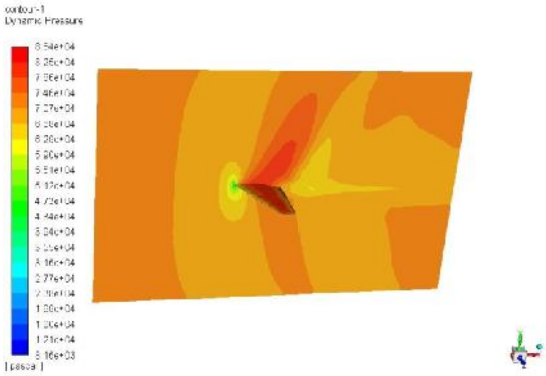


Fig.10 Dynamic pressure at M=1 at5 ^{0}AOA

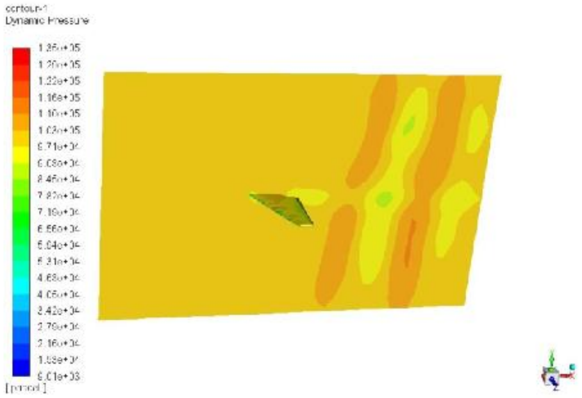


Fig.11 Dynamic pressure at M=1.2 at 10AOA

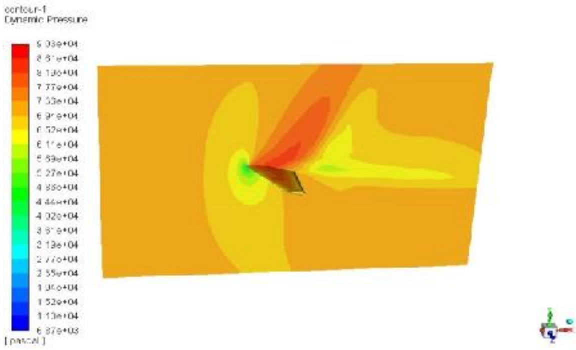


Fig.12 Dynamic pressure at M=1.5 at5^{0}AOA

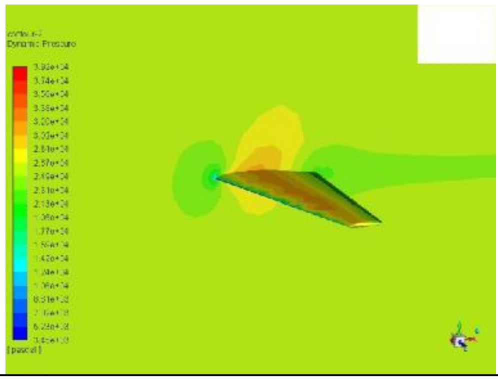


Fig.13 Dynamic pressure at M=0.6 at 0^{0}AOA

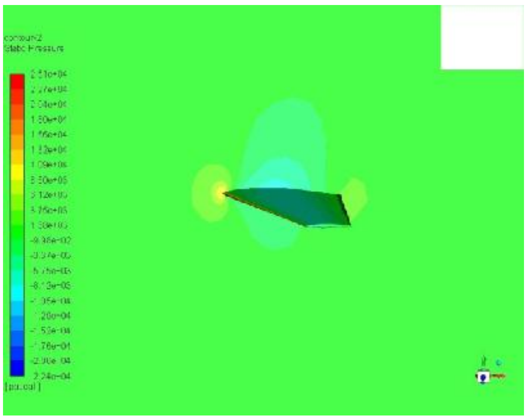


Fig.14 Static pressure at M=0.6 at 0^{0}AOA

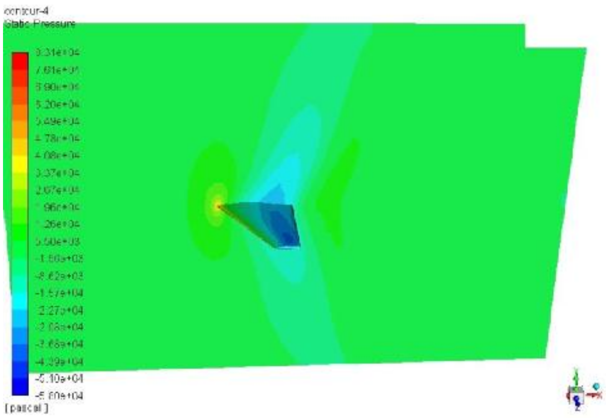


Fig.15 Static pressure at M=1.2 at 5^{0}AOA

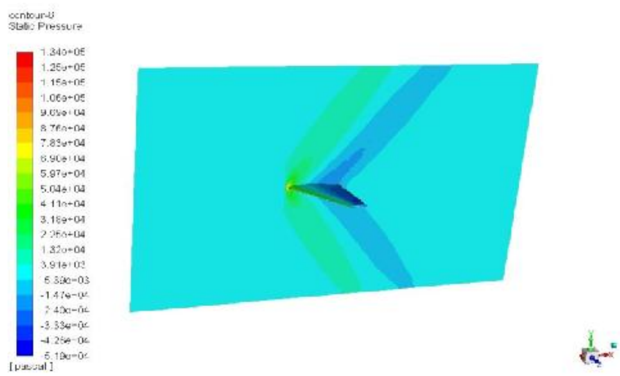


Fig.16 Static pressure at M=1.5 at 5^0AOA

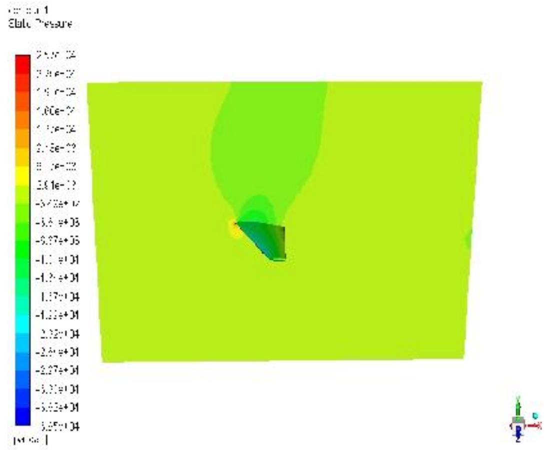


Fig.17 Static pressure at M=0.6 at 10^0AOA

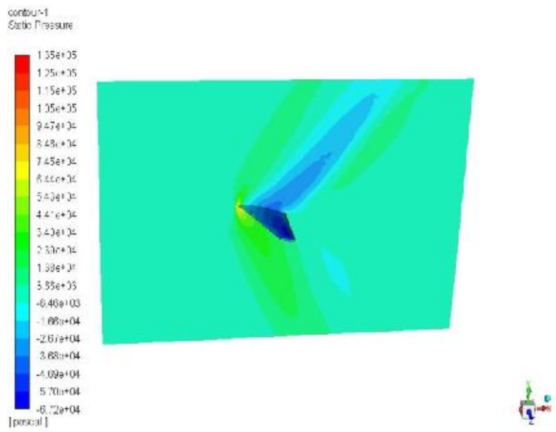


Fig.18 Static pressure at M=1.2 at 15^0AOA

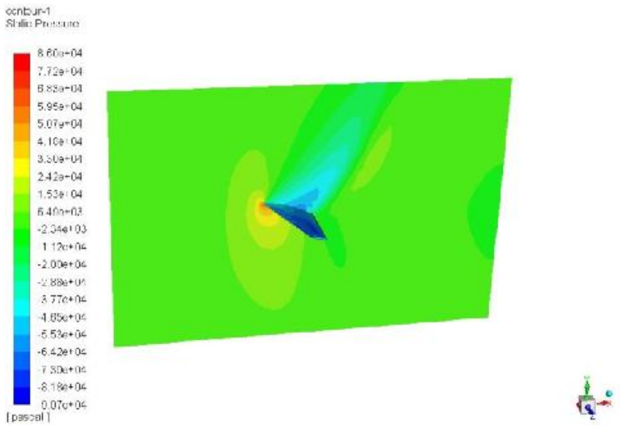


Fig.19 Static pressure at M=1.2 at 15^0AOA

Results and Discussions:

From the statistical data, the results obtained from computational analysis, that is coefficient of lift is maximum at Mach number 1.5 at 15 degree angle of attack. That is for F-16 highest Mach number and lower angle of attacks the aerodynamic efficiencies will be comparatively effective**[4]** whereF-16 requires minimum thrust to operate **.** From the results we can seen that the aerodynamic efficiency will be higher at 0^0angle of attack for M=0.6, the results of F-16 aircraft analysis are carried out and the data is collectively compared for solutions [8-11]

It is the highest-Mach-number case and the lowest-angle-of-attack case for F-16 falcon where it has to obtain the higher speeds where aerodynamic efficiency is maximum. Here from the results obtained are comparatively satisfying the conditions that is coefficient of lift is maximum at higher Mach number 1.5 at 15 degree angle of attack that is lower angle of attack case where aerodynamic efficiency is higher. The occurrence of a shock–vortex interaction may be unsteady not studied here; the significance changed the surface pressure distribution. The results can be interpreted as rise in velocity magnitude and reductions in the static pressure as for lower angles of attack the flow will not separate much over the wing. But when the angle

International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 4, Issue 11
November 2017

of attack increases the flow will slightly separate from the upper surface of the wing where coefficient of drag will be at the maximum. At Mach number 0.6, formation of two large leading edge vortices which is the subsonic lift generation the suction effect of the leading edge vortices increase the normal force which will enhances the lift.

Conclusions:

From this we can conclude that the aerodynamic efficiency of the modified delta wing here considering F-16 wing reference gives effective results at higher mach speeds at lower angle of attacks. For an fighter aircrafts the CFD results should give effective results at both subsonic and supersonic regions at high speed at normal altitudes. So, as the Mach number increases the there will be increase in drag. The modified design of the airfoil considered for the modified F-16 wing results in effective increase in aerodynamic efficiency at higher Mach numbers at lower angles.

References

1. Dean, John P., Scott A. Morton, David R. McDaniel, James D. Clifton, and David J. Bodkin. "Aircraft stability and control characteristics determined by system identification of CFD simulations." In *AIAA Atmospheric Flight Mechanics Conference and Exhibit*, pp. 18-21. 2008.
2. Talty, Patrick K., and Donald J. Caughlin. "F-16XL demonstrates new capabilities in flight test at Edwards Air Force Base." *Journal of Aircraft* 25, no. 3 (1988): 206-215.
3. Good heart, Kevin A., and Günter H. Schnerr. "Condensation on ONERA M6 and F-16 wings in atmospheric flight: Numerical modeling." *Journal of aircraft* 42, no. 2 (2005): 402-412.
4. Rizzi, Arthur, Adam Jirásek, John E. Lamar, Simone Crippa, Kenneth J. Badcock, and Okko J. Boelens. Lessons learned from numerical simulations of aircraft F-16XL aircraft at flight conditions, *Journal of Aircraft* 46, no. 2 (2009): 423.
5. Dean, John P., James D. Clifton, David J. Bodkin, Scott A. Morton, and David R. McDaniel. "Determining the applicability and effectiveness of current CFD methods in store certification activities." *AIAA paper* 1231 (2010).
6. Clifton, James D., C. Justin Ratcliff, David J. Bodkin, and John P. Dean. "Determining the stability and control characteristics of high-performance maneuvering aircraft using high-resolution CFD simulation with and without moving control surfaces." *AIAA Paper 2012* 4656 (2013).
7. Jameson, Antony. "The present status, challenges, and future developments in computational fluid dynamics." In *AGARD CONFERENCE PROCEEDINGS AGARD CP*, pp. 1-1. AGARD, 1996.
8. Peiris, H. A. M., P. V. S. Nirmal, H. M. H. H. S. Bandara, D. M. Mahindarathne, and SLMD Rangajeevaand RMPS Bandara. "Aerodynamics Analysis of F-16 Aircraft."
9. Sadraey, Mohammad. "Wing Design." *Sep-2012.[Online]. Available: http://faculty. dwc. edu/sadraey* (2012).
10. Luckring, James M. "A survey of factors affecting blunt leading-edge separation for swept and semi-slender wings." In *28th Applied Aerodynamics Conference, AIAA*, vol. 4820, p. 2010. 2010.
11. Canpolat, C., S. Yayla, B. Sahin, and H. Akilli. "Observation of the vortical flow over a yawed delta wing." *Journal of Aerospace Engineering* 25, no. 4 (2011): 613-626.