International Journal of Research in Advent Technology (E-ISSN: 2321-9637) Special Issue
National Conference "IAEISDISE 2014", 12-13 September 2014

Numerical Simulation of Civil Aircraft Fuselage Using CFD Technique

Srinivas G[1*], Srinivasa Rao Potti[2], Chethan K N[3], K K Arvind Kumar[4]
Asst.Prof, Department of Aero & Auto Engg, MIT-Manipal University, Manipal, Karnataka – 576104[1,3]
Assoc Prof, Department of Hum & Mangt Engg, MIT-Manipal University, Manipal, Karnataka -576104[2]
Asst Prof, Department of Aeronautical Engg, MRCET-Hydeabad – 500014[4].
*Email: srinivasle06@gmail.com1**

Abstract- Nowadays the design of fuselage is playing very important role in the field of aircraft industry. To enhance the aerodynamic efficiency, reducing the drag effect on the surface of the fuselage body still developments is going on in both experimental and theoretical as well. Here the Numerical simulation plays important role to reduce the experimental cost and errors. This paper deals with Numerical simulation flows around civil aircraft fuselage was conducted to investigate the aerodynamic interaction of main body and other components such as control device and tail. For this determination, a three-dimensional in viscid flow solver has been established based on unstructured meshes. An overset mesh technique was used to describe the relative motion between the main body and other components too. As the application of the present method, calculations were made for the fuselage aerodynamic interaction. Comparison of the computational results was made with measured time-averaged and instantaneous fuselage surface pressure distributions and thrust distribution and available experimental data. It is demonstrated that the present method is efficient and robust for the simulation of complete civil aircraft configurations.

Index Terms- Design; Aircraft; Fuselage, CFD, Aerodynamics.

1. INTRODUCTION

The flow field and aerodynamic performance of fuselage have an important impact on the performance of a aircraft, such as flight quality, acoustic and vibration characteristics[1,6]. Accurate calculation of rotor flow field and the performance analyses with computational fluid dynamics (CFD) method are important directions of the aircraft aerodynamics. Due to the complex configuration of a aircraft, there must be drastic aerodynamic interactions between the main fuselage body, tail, and further engine inlet and outlet[3,4]. Hence, it is of great significance to the prediction of these interactions. In spite of its importance, aircraft aerodynamics interactions have little been fully studied due to numerous reasons, like complex vortex evolution and mechanisms and rotor wake geometry[2,5]. Although interactions between the main rotor, tail rotor and fuselage have been studied by many researchers, the engine inlets and outlets influences are usually ignored. However, the existence of the engine inlets and outlets may have a tremendous impact on the rotor flow field, since they are major heat sources of a helicopter. So it is significant to investigate interactions between them. What is more, interactions between the actual rotor motion and slung loads of specific geometries have so far been little considered.

Unlike the previously considered fuselage / airfoil shapes which have been developed to be used as wings or blades, the fuselage/airfoil has been developed with the objective of deploying it as a fuselage of an aircraft. This entails a tradeoff between various conflicting requirements; firstly the thickness has to be adequate to create enough cubical volume within the contours of the fuselage/airfoil to accommodate the desired passenger and cargo capacities. Also the aerodynamic essentials viz: maximum lift and minimum drag have to be strived for along with ergonomic and architectural requirements. The provision for accommodating flight deck as well as avionics and control systems has also to be made. The fuselage/airfoil so developed meeting the conflicting requirements has been designated as HLF(High Lift Generating Fuselage) aerofoil. This high lift generating airfoil was numerically analyzed in a similar way as described earlier in case of other fuselage/airfoils.

Thus the present results being in reasonable agreement with each other for all practical purpose the CFD results and the methodology adopted as well as the boundary conditions employed may be treated as valid.

2. METHODOLOGY

In this paper, the most commonly used finite volume method in CFD is used. Actuator disc model for main fuselage has been widely; hence it is also adopted by this paper to model the downwash of aircraft fuselage. The engine inlets and outlets are also in consideration. As the study is only about the flow field around the fuselage engine inlets and outlets, there is no need to simulate the interior flow field of the engine. Engine inlets and outlets are set as boundary conditions with given parameters like speed, flux and pressure.

Governing equations and turbulence model

Computational fluid dynamics, usually shortened as CFD, is the analysis of systems linking fluid flow, heat transfer and related phenomena such as chemical reaction by means of computer-based simulation. The method is very powerful and spans a wide range of industrial and non-industrial solicitation area. Some of the illustrations are: Aerodynamics of aircraft and vehicles: lift and drag, Hydrodynamics of ships, Power plant: combustion in IC engine and gas turbines, Chemical process engineering: mixing and separation, polymer molding, External and internal environment of buildings: wind loading and space heating/ventilation

Since past aerospace industry has combined CFD techniques into the design, R&D and building of aircraft and jet engines. More recently the methods have been functional to the design of internal combustion engines, combustion chambers of gas turbines and furnaces. Likewise, motor vehicle manufacturers now routinely predict drag forces, under-bonnet air flows and the in-care situation with CFD. Increasingly FCD is attractive a vital component in the design of industrial products and procedures. A CFD code works in following steps like Pre-Processor, Solver, Post-Processor.
The governing equations of fluid flow and heat transfer are: Continuity equation: The corporeal principle states the mass of the fluid is conserved. It can neither be created nor be destroyed. The final result of applying the physical principle of the conservation of mass to a finite control volume fixed in space. This equation is called the continuity equation.

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \tag{1}$$

Momentum equation: The physical principle states that the rate of alteration of momentum equals the sum of the forces on the fluid particle (Newton's second law).
X-component momentum

$$\rho\frac{Du}{Dt}=\frac{\partial(-p+\tau_{xx})}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+S_{Mx} \tag{2}$$

Y-component momentum

$$\rho\frac{Dv}{Dt}=\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial(-p+\tau_{yy})}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+S_{My} \tag{3}$$

Z-component momentum

$$\rho\frac{Dw}{Dt}=\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial(-p+\tau_{zz})}{\partial z}+S_{Mz} \tag{4}$$

Energy equation:The physical principle states that the amount of change of energy is equal to the sum of the rate of heat addition to and the rate of work done on the fluid particle (first law of thermodynamics).

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla \tag{5}$$

$$\bullet\left[\rho\left(e+\frac{V^2}{2}\right)V\right]=\rho\dot{q}-\nabla\bullet(\rho V)+\rho(f\bullet V)+\dot{Q}'_{viscous}+\dot{W}'_{viscous}$$

Many turbulence models have been developed to relate the fluctuating amount and the average amount, so that the governing equations of the average motion can be solved. The complete flow analysis can see from figure 1-10. The comparisons can see figure 11-13 for difference Reynolds numbers.

Flying altitude	6000m
Advance Ratio	0.05
Fuselage front radius	10.65m
Weight	144,1400kg
Angle of attack	4^0

Table 1: Boundary conditions

ANGLE OF ATTACK	CFD DATA	
	CL	Cd
0	0.70761	0.00167868
2	0.9587	0.0023499
4	1.2054	0.0033566
6	1.4521	0.0044298
8	1.6708	0.0067398
10	1.9035	0.006779
12	2.107	0.0082801
14	2.2762	0.010381

Table 2: Variation of AOA with coefficients

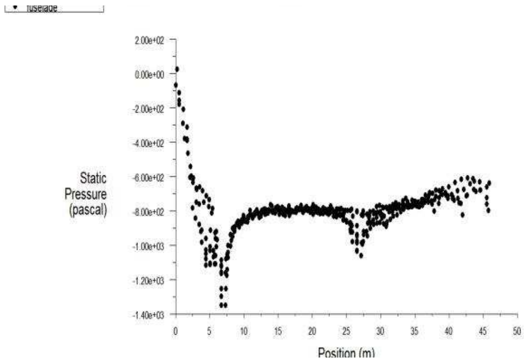

Figure 1: Static Pressure along the fuselage

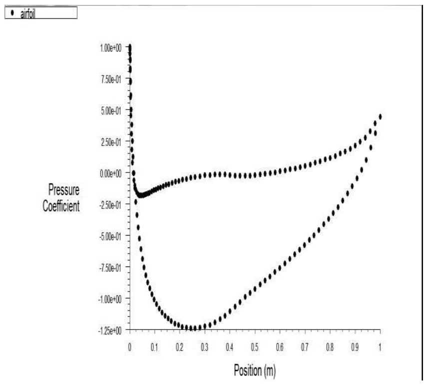

Figure 2: Pressure coefficient along the fuselage

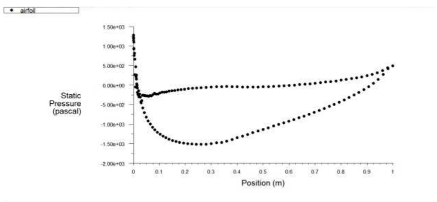

Figure 3: Static Pressure along the fuselage axis

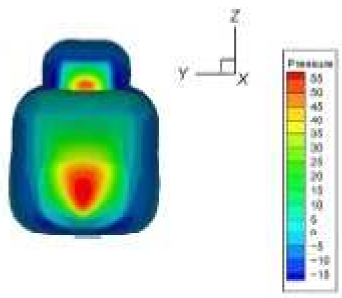

Figure 4: Fuselage front view pressure distribution

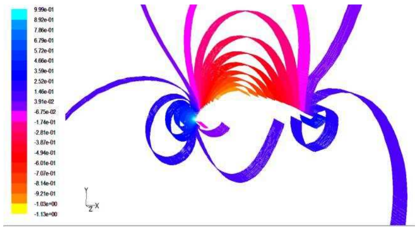

Figure 5: Pressure contour around the fuselage

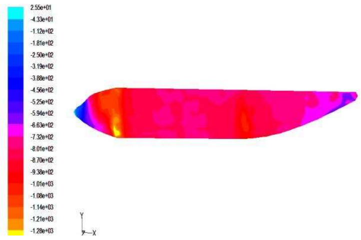

Figure 6: surface Pressure around the fuselage

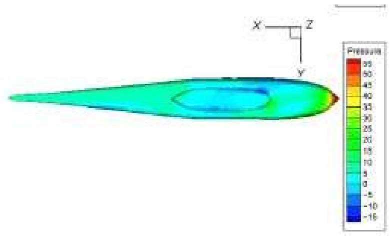

Figure 7: Top view pressure distribution of the fuselage

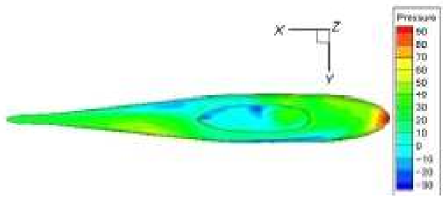

Figure 8: Pressure distribution of A380 Top view

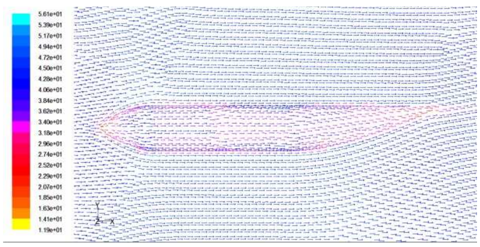

Figure 9: Velocity contour along the fuselage side

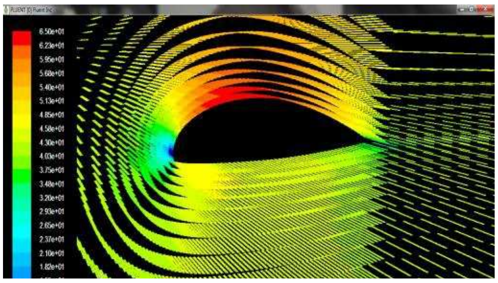

Figure 10: Velocity contour around the fuselage

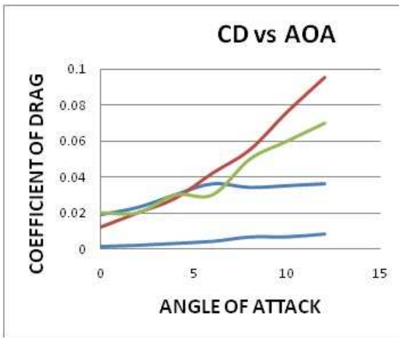

Figure 11: CD verses AOA

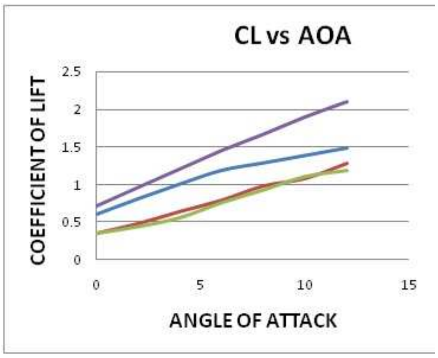

Figure 12: C_L verses AOA

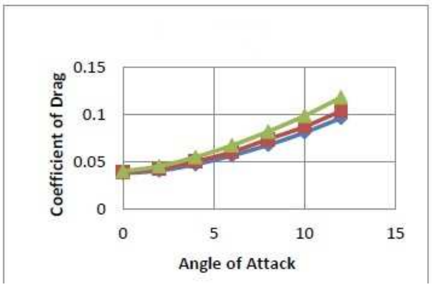

Figure 13: Comparison Graph

3. RESULTS AND DISCSUSSION

The drag reduction in turn results in reduced takeoff and landing velocities of the aircraft, significantly reducing their propulsive thrust and fuel burn requirements. This design has tremendous potentials not only in terms of improved economy and reduced emissions, but also enhanced safety of operation. The initial calculations and estimations were done by obtaining the technical data of three of the most commonly used passenger aircrafts viz, Boeing 787, Airbus A380, and Boeing 707. The modified fuselage shapes were based on some of the standard published NASA aerofoils, and using the available data, the implications of the modified fuselage shape were evaluated. Here propulsive thrust and fuel burn requirements. This design has tremendous potentials not only in terms of improved economy and reduced emissions, but also enhanced safety of operation.

To perform the numerical analysis the flow field around the fuselage was modelled using GAMBIT software. While doing the meshing the following criteria were kept in mind. The CFD analysis of fuselage is done in two stages. First the two dimensional fuselage profile is analysed and compared with the three dimensional fuselage. The subsequent results were then compared with the Airbus A380 fuselage. The comparison of the data arrived at by the CFD approach is given in the tabular form for fuselage and Airbus A380 fuselage.

Also the aerodynamic essentials viz: maximum lift and minimum drag have to be strived for along with ergonomic and architectural requirements. The provision for accommodating flight deck as well as avionics and control systems has also to be made. The salient features of the fuselage cross section so developed meeting the conflicting requirements are as below:
i. Maximum Camber: 15%
ii. Maximum Camber Location: 60% from LE
iii. Maximum Thickness (Upper): 16%
iv. Maximum Thickness (Lower): 8%
v. Location of Maximum Thickness. (Upper): 25% from LE
vi. Location of Maximum Thickness (Lower): 46% from LE
vii. Trailing Edge Thickness: 5%
viii. Overall Maximum Thickness: 15%

The special fuselage developed on the contours of the fuselage so as to accord the same cubical space within its walls as the Airbus A380 fuselage would have the following salient dimensions.
i. Chord Length = 54 meters
ii. Width Span = 5.8 meters

The smooth pressure contours and chord wise pressure distribution curves also explain the superiority of the aerodynamics of the airfoil and hence its performance. This is natural as the aerofoil has been specially developed for the application.

4. CONCLUSIONS

To, conclude the proposed high lift generating Fuselage, synthetically, has the ability to be deployed as an aircraft fuselage. Its rather large and uniform thickness over the significant portion of the chord imparts much sought cubical space within the contour of the airfoil. This makes it the ideal choice for the fuselage application to achieve short landing and takeoff (STOL) capabilities of aircraft. Further it is worth to mention that the 3D Fuselage does not show any flow separation in the investigated range of AOA i.e. from 0^0 to 12^0 and does not give rise to any stalling condition in this range of AOA. It is obvious from the foregoing results that the 3D Fuselage has much superior lift characteristics coupled with low drag characteristics especially at lower angles of attack. Quantitatively, in cruise conditions i.e at zero degrees AOA the Lift coefficient of 3D Airbus A380 Fuselage is approximately 0.2 – 0.5% of lift produced by 3D Fuselage.

REFERENCES

[1] Storaasli, Olaf L., "Correlation of Finite Element Prediction and Experimental Measurments of Aircraft Panel Structural Acoustics," M.S. Thesis, Department of Aerospace Engineering, Old Dominion University, 1997.

[2] C.Kindervater; Aircraft and Helicopter Crashworthiness-Design and Simulation, "Crashworthiness of Transportation Systems" ,Kluwer Academic Publishers, (1997).

[3] K.Kondo,et al; Investigation Report on the Methods of Tests and Numerical Simulation on the Crashworthiness of Full-Scale Aircraft Structures, Foundation for Promotion of Japan Aerospace Technology(2001,In Japanese).

[4] I.Kumakura, H.Terada; Research Plan at NAL on Drop Test of Fuselage Structure of YS-11 Turbo-prop Transport Aircraft, The 3rd Int. Aircraft Fire and Cabin Safety Research Conference (Atlantic City ,USA, 2001)

[5] F. Le Chuiton, A. D"Alascio, G. Barakos et al, "Computation of the Helicopter Fuselage Wake with the SST, SAS, DES and XLES Models", Springer eBook, January 24, 2008.

[6] Mueller, T. J., (editor), "Conference on Fixed, Flapping and Rotary Wing Vehicles at Low Reynolds Numbers," Notre Dame University, Indiana, June 5-7, 2000.