CFD TOOLS AS APPLIED TO INDIAN COMBAT AIRCRAFT CONFIGURATION-AN EXPERIENCE

K.P. Singh*

Abstract

Computational Fluid Dynamics (CFD) tools are extensively used by aircraft industries all over the world in design and development of airplanes, with emphasis on reducing both the cost and design cycle time. This paper deals with the use of CFD tools at Aeronautical Development Agency (ADA) in design and development of Indian combat aircraft. The applications include: (1) evaluation of incremental changes in aerodynamic configuration, (2) wave drag evaluation, (3) aerodynamic loads, (4) flow field around air data sensor locations, (5) Euler calculation for coupled internal and external flow field, and (6) viscous computation of high angles of attack flows. In the initial design phase, codes based on the potential theory were used for wing design and fuselage shape optimization. Later indigenously developed Euler and Navier-Stokes codes were employed for the various applications. This paper brings out the strength and weakness of CFD as experienced at ADA and its comparison with the Wind Tunnel and Flight Data.

Introduction

CFD tools are extensively used by aircraft industries all over the world for design and development of airplanes with emphasis on reducing both the cost and design cycle time. Alternative to CFD analysis is wind tunnel testing which is expensive as well as time consuming. However, wind tunnel testing is very essential in certain areas such as estimation of aerodynamic coefficients and determination of intake buzz boundary etc. On the other hand, there are many areas where CFD had played very important role and become an economical supplement to wind-tunnel testing in reducing the cost and time of design cycles.

Aeronautical Development Agency (ADA) has taken up the task of design and development of a modern combat aircraft. A tail-less delta configuration, as shown in Fig.1, is considered a good candidate for a combat aircraft with desirable maneuverability and acceleration characteristics. In order to improve its performance further, leading edge slats (LES) have been employed to delay leading edge flow separation.

CFD has contributed significantly in diverse areas in design and development of Indian combat aircraft. During the Preliminary Design Phase (PDP), the codes were extensively used for studying various configuration options and freezing the ideal candidate. For this purpose, CFD codes based on linear and full potential theory were used as they provide reasonably accurate design data with less computer time.

During the Development Phase, higher order CFD codes were used. In this phase of design, incremental changes in aerodynamic coefficients due to minor changes in configuration were studied. Study was also carried out to evaluate the optimal deflection of leading edge devices. CFD was immensely used to compute aerodynamic load distribution over various components of aircraft such as radome, fuselage, wing, stores, fin, etc. is required for structural design of these components. The aerodynamic loads on the control surfaces and leading edge devices are also computed for selection of the appropriate size of the actuators. The other equally important requirements accomplished through CFD include calculation of control surface hinge moments, load sharing, preliminary calibration data for air-data sensors and flow field studies inside the intake duct. CFD has also contributed in studying the safe separation of stores from parent aircraft. The last, but not the least, is the visualization of flow field over the entire aircraft which has helped the designers to study the mutual interaction between various components of the aircraft.

In this paper, the contribution of CFD in design and development of a modern combat aircraft at Aeronautical Development Agency (ADA) is presented in a few se-

* Group Director, Aeronautical Development Agency, Vimanapura Post, Bangalore-560 017, India
lected areas, namely, (i) evaluation of incremental changes in aerodynamic configuration (ii) wave drag evaluation (iii) aerodynamic loads (iv) flow field around air data sensors (v) Euler calculation for coupled internal and external flow field and (vi) computation of high angles of attack flows. This paper brings out the strengths and weakness of CFD as experienced at ADA and its comparison with the Wind Tunnel and Fight Data.

Evaluation of Incremental Changes in Aerodynamics Configuration

After freezing the configuration (Config-A) of an aircraft during PDP, some minor geometric changes may often be required to improve its performance. It would be a time consuming task if each change is to be further evaluated through wind tunnel testing. For this purpose, CFD tools are found to be reliable and also very economical in terms of cost and time. In the case of present combat aircraft, the design team during the development phase wanted studies on effect of backward shift of wing, forward shift of intake duct entry and an increase in fuselage length (Config-B). The designers were interested to assess the changes in stability margin of the aircraft due to these changes in the configuration. The study was carried out at transonic Mach numbers using an in-house developed full potential code FETRAN (Finite Element Transonic code) [1].

Table-1 shows the comparison of incremental changes in stability margin, evaluated with CFD and wind tunnel testing. It can be seen from this table that even though the accuracy in predicting the stability margin is about 15% in absolute value, the incremental changes in stability has been predicted within the accuracy of 2%. It is worthwhile to note here that the wind tunnel test data generated subsequently confirmed the CFD values provided ahead of the tests.

Wave Drag Evaluation

CFD tools are found to be very useful to evaluate wave drag increase due to change in configuration. When aircraft configuration was being modified to trainer configuration, the change in canopy shape, as shown in Fig. 2, was evaluated for wave drag increase using HYENA - an in-house developed Euler code [2]. With the feedback from CFD analysis, wave drag was reduced by 50% [3]. Further, the wave drag increase was also computed due to nose cone droop employed for better vision angle (Fig.2). The computed wave drag [4] due to both the changes was later compared with the experimental data [5] and found about 10% lower. This difference may be attributed to the fact that experimental data was generated with some of the aerodynamic fixes which were not considered in the CFD studies.

Wave drag was also computed for aircraft configuration and compared with both the experimental value and flight data. The computed drag was under predicted by 3% and 10% as compared to experimental and flight data [6] respectively.

Aerodynamic Loads

CFD tools based on potential and Euler equations have been used extensively to compute aerodynamic loads on various components of the aircraft such as radome, canopy, fuselage, wings, fins, actuator fairings etc. However, global aerodynamic force and moment coefficients have been taken from the wind tunnel testing. The comparison of pressure distribution obtained with CFD and the experimental data from National Aerospace laboratories (NAL) wind tunnel (WT) over the wing for M= 0.95 and the angle of attack (AOA) of 4.65 and 9.416 degrees is shown in Figs. 3 and 4 at two spanwise stations. The results are obtained using AMES - an indigenously developed Euler code [7]. The comparison is very good. The effect of actuator fairing at the bottom surface of the wing is well captured by the Euler code. For the purpose of structural analysis, it is convenient to represent the pressure distribution (Cp) in the form of dCpα (Variation of dCp - pressure (Cp) difference between bottom and top surfaces of wing with angle of attack ). Fig.5 shows the comparison of dCpα over the wing with the data obtained from AMES and FIM (potential) codes at M=0.95. As can be seen from this figure, FIM code fails to capture the trend in dCpα near the trailing edge while agreement between Euler code and the test data is good.

| Table-1 : Comparison of incremental changes in stability margin |
|-------------|-------------|--------|
| Configuration (A) | Configuration (B) | A-B  |
| Experimental | 2.48 | -1.55 | 4.03 |
| CFD | 2.10 | -1.86 | 3.96 |
| Error | 15% | 16% | 1.7% |
Currently, the Cartesian grid based Euler solvers - CARGIL and PARAS are being used for CFD analysis of the aircraft with all the heavy stores configurations. These solvers are very effective because of less turn around time in grid generation process.

Flow Field Computation Around Air Data Sensor Location

Towards providing adequate redundancy, Air Data System (ADS) of Indian combat aircraft has been configured with five sensors location on front fuselage to measure total and static pressures and local flow angularity. The sensors include two Side mounted Air Data Probes (SADP), two Angle Of Attack Vanes (AOAV) and one Nose Air Data Probe (NADP) (Fig.6). The Euler code-AMES was used to generate local flow field (static pressure and flow angularity) data over the aircraft front fuselage. Flow field data thus obtained was used for two purposes. First, to find a suitable location on the front fuselage for SADP and AOAV at which the local flow field is sensitive to the free stream angles of attack and the side slip angles. Secondly, to compute the static pressure and local flow angularity at the selected locations of the SADP and AOAV. A typical comparison with experimental values [8] at M=0.7 is shown in Fig. 7. More details of comparison can be found in Ref. [9]. Fig.8 shows a comparison between the computed values and the flight data at M=0.4 [10].

These validated CFD procedures have been very useful to compute the local flow field at the new location of air data sensors on the front fuselage of Trainer/Navy version of Indian combat aircraft.

Internal-External Flow Computation

It is very important to understand the nature of flow in front of the intake duct. The flow is very complex due to the presence of splitter plate and the top and bottom diverters. Shielding of intake with the wing, as shown in Fig. 9, makes the flow even more complex. To handle this type of complex geometry, an in-house developed unstructured Navier-Stokes code - VISIP3D [11] has been employed for the flow field calculation. The unstructured grid generated over the surface is shown in Fig. 9. Calculations have been carried out at a free-stream Mach number of 1.4 for two mass flow conditions of $S^* = 15.32$ and 22.10 respectively. Only Euler solution is computed in the present study [12]. Figs.10 and 11 shows the comparison of pressure variation with the experimental data [13] in the region ahead of intake duct at the two generators. It can be seen that shock location is predicted reasonably well. However, the computed pressure peaks through shocks are over predicted. This may be due to the viscous effects.

Computations at High Angle of Attack

Viscous computations over the combat aircraft configuration are carried out employing the CNS3D - an in-house developed RANS code [14,15] based on block structured grids. The computational grid is composed of 1.5 million mesh points. Calculations are performed at a free stream Mach number of 0.7 and the angles of attack of 13.272 and 17.714 degrees respectively. Variation of coefficient of pressure for $\alpha = 13.272$ and 17.714 degrees at two stations of 35% and 48% semi-span is shown in Figs.12 and 13 respectively. Predictions have been compared with the experimental data from NAL wind tunnel and a fairly good comparison is observed. The vortical flow field captured in calculations is presented in Fig. 14. The particle traces illustrate the detailed structure of the rolled-up vortices. It is seen that the leading edge vortices are strengthened as the flow develops progressively away from the leading-edge of the wing.

Conclusion

CFD tools have been used extensively for the design and development of Indian combat aircraft at ADA. They have contributed significantly in the area of preliminary design, assessment of incremental changes in geometry, wave drag evaluation, flow field data around the air data sensors, aerodynamic loads. CFD has been proved very handy and reliable tool in these areas. CFD is also helping in understanding the flow structure at high angles of attack and coupled internal-external flow. From the experience at ADA, it is worth mentioning that the CFD inputs have helped the designers in reducing both the cost and time in aircraft development programme. CFD would still take some more time to contribute significantly in areas such as high angle of attack aerodynamics, prediction of intake buzz, unsteady aerodynamics, etc.

Acknowledgement

Author wishes to acknowledge the inputs and help received from his colleagues in preparation of this manuscript.

References

1. Nandanan, M., "FETRAN - A Computational Fluid Dynamics Software for Analysis of Aircraft Con-


Fig.1 Top and side view of tail less delta aircraft

Fig.2 Air Force configuration (left), Navy/Trainer configuration (right)
Fig. 3 Variation of Cp over wing surface at 35% (left) and 48% (right) spanwise stations at M=0.95 and AOA=4.65 degree

Fig. 4 Variation of Cp over wing surface at 35% (left) and 48% (right) spanwise stations at M=0.95 and AOA=9.416 degree

Fig. 5 Unit aerodynamic load variation over aircraft wing at M=0.95
Fig. 6 Location of SADP and AOA vane on front fuselage

Fig. 7 Comparison of CFD prediction with experimental data at SADP location: Flow angle (left), static pressure (right)

Fig. 8 Comparison of CFD prediction with flight data at: flow angle at vane location (left), static pressure at SADP (right)
Fig. 9  Pressure port locations (left), surface grid in intake region (right)

Fig. 10  Pressure distribution for $S^* = 15.32$ and 22.01 at Section B

Fig. 11  Pressure distribution for $S^* = 15.32$ and 22.01 at Section D
Fig. 12  $C_p$ variation at 35% and 48% semi-span stations, respectively, $M_\infty=0.7$, $\alpha=13.272$ deg., $\beta=0$ deg., $Re=87.09x10^6$ based on MAC of aircraft.

Fig. 13  $C_p$ variation at 35% and 48% semi-span stations, respectively, $M_\infty=0.7$, $\alpha=17.714$ deg., $\beta=0$ deg., $Re=87.09x10^6$ based on MAC of aircraft.

Fig. 14  Vortical flow structure at $M_\infty=0.7$, $\alpha=17.714$ deg., $\beta=0$ deg., $Re=87.09x10^6$ based on MAC of aircraft.