Airplane Performance Analysis through CFD Results: Assessment of a NACA 64A212 Wing-body Combination

JI Abeygoonewardena[1], RMPS Bandara[2]

[12]General Sir John Kotelawala Defence University, Sri Lanka
[1] jabeygoonewardene@kdu.ac.lk, [2] bandara@kdu.ac.lk

Abstract *- A comprehensive understanding of fluid and flight dynamics of an aircraft cannot be understated from both an engineering and aircraft handling perspective. In this context, availability of such descriptive information, especially in the case of trainer aircraft, will deliver profitable results for both students and trainers. Detailed information regarding the aerodynamic behaviour of the wing is not disclosed by the aircraft manufacturer to relevant stakeholders. A research was conducted by the same researchers to examine the aerodynamic behaviour of an aircraft having a wing tip designed with the NACA 64A212 airfoil through Computational Fluid Dynamics (CFD) approach and a generic lift curve and drag polar was developed for flows of reasonable Reynolds number.*

Present work is involved with the assessment of performance of the NACA 64A212 wing-body combination based on the results obtained for the 2-D airfoil. Solid modelling software is used to create the geometry and computational mesh of the wing-body combination. A CFD tool models the flow physics involved in flight, thus rendering performance parameters which are available only through trial and error in the present context. The results provide new insights into the behaviour of the wing-body combination, thus enabling means of enhancing performance and handling qualities of the aircraft for both designers and pilots.

Keywords - Aircraft Performance, CFD, Handling Qualities

I. INTRODUCTION

Solutions of the complete Navier-Stokes equations for flow fields over two-dimensional and three dimensional bodies have been presented in numerous work, as it is an underlying prerequisite for the design of flight vehicles. Modelling of flow physics and comprehensive understanding of the aerodynamics of various surfaces of a flight vehicle, including the wing-body combination as a whole is of paramount importance for both the designer and handler. Basic aerodynamic behaviour of the NACA 64A212 airfoil in the context of Computational Fluid Dynamics (CFD) was analysed by Bandara & Abeygoonewardene, (2013) for the two dimensional case. The two dimensional flow for subsonic and transonic flow regimes were observed. The work presented in this paper is part of an effort in developing a complete model for the flow physics of a three dimensional wing-body combination for an aircraft having a NACA 64A212 airfoil tip and NACA 64A114 airfoil root. There is no accurate analytical equation which can predict the lift of a wing-body combination due aerodynamic interactions of the wing and main fuselage. The approach is either the configuration is to be tested in a wind tunnel or a CFD analysis is to be conducted. A time dependent solution for the Navier-Stokes equation was conducted by Shang & Scherr, (1986) for the hypersonic viscous flow over the X-24C hypersonic test vehicle.

The Navier-Stokes Equations are given below in the three dimensional unsteady form:

Continuity:

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

X-Momentum:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{1}{\mathrm{Re}_f}\left[\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z}\right] \rightarrow (2.1)$$

Y-Momentum:

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial p}{\partial y} + \frac{1}{\mathrm{Re}_f}\left[\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z}\right] \rightarrow (2.2)$$

Z-Momentum:

$$\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uw)}{\partial x} + \frac{\partial(\rho vw)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial p}{\partial z} + \frac{1}{\mathrm{Re}_f}\left[\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z}\right] \rightarrow (2.3)$$

$$\frac{\partial(E_T)}{\partial t}+\frac{\partial(uE_T)}{\partial x}+\frac{\partial(vE_T)}{\partial y}+\frac{\partial(wE_T)}{\partial z}$$
$$=-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}$$

$$-\frac{1}{\mathrm{Re}_f\,\mathrm{Pr}_f}\left[\frac{\partial q_x}{\partial x}+\frac{\partial q_y}{\partial y}+\frac{\partial q_z}{\partial z}\right]$$

$$+\frac{1}{\mathrm{Re}_f}[\frac{\partial}{\partial x}\left(u\tau_{xx}+v\tau_{xy}+w\tau_{xz}\right)$$
$$+\frac{\partial}{\partial y}\left(u\tau_{xy}+v\tau_{yy}+w\tau_{yz}\right)+\frac{\partial}{\partial z}\left(u\tau_{xz}+v\tau_{yz}+w\tau_{zz}\right)]$$
$$\rightarrow(3)$$

The said equations consist of a time-dependent continuity equation for the conservation, three time dependent conservation of momentum equations and a time-dependent conservation equation for energy. The spatial coordinates of the domain, denoted by **x**, **y** and **z**, as well as the time, t, are the four independent variables. The six dependent variables include pressure, density, temperature and the three components of the velocity vector; **u**, **v**, and **w** in the **x**, **y** and **z** directions respectively. All dependent variables are functions of all independent variables, thus resulting in a set of partial differential equations.

In investigating transonic and supersonic behaviour of flow over an entire wing, where entropy gradients are present, thereby making the flow rotational; the Navier-Stokes equations are employed in the following form with no body forces:

Continuity:

$$\frac{\partial\rho}{\partial t}=-\left(\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}\right)$$

X- momentum:

$$\frac{\partial u}{\partial t}=-\left(u\frac{\partial(u)}{\partial x}+v\frac{\partial(u)}{\partial y}+\frac{1}{\rho}\frac{\partial(p)}{\partial x}\right)$$

Y-momentum:

$$\frac{\partial v}{\partial t}=-\left(u\frac{\partial(v)}{\partial x}+v\frac{\partial(v)}{\partial y}+\frac{1}{\rho}\frac{\partial(p)}{\partial y}\right)$$

Energy:

$$\frac{\partial(e+V^2/2)}{\partial t}=-\left(\begin{array}{l}u\frac{\partial(e+V^2/2)}{\partial x}+v\frac{\partial(e+V^2/2)}{\partial y}+\frac{1}{\rho}\frac{\partial(pu)}{\partial x}\\+\frac{1}{\rho}\frac{\partial(pv)}{\partial y}\end{array}\right)$$

Thus, these equations provide the form necessary for a time-dependent finite difference solution.

II. METHODOLOGY AND APPROACH

Performance assessment of the aerodynamic behaviour of the aircraft wing-body combination was conducted using the following tools:

- Geometrical/Solid modelling tool
- Mesh generation tool
- Computational fluid dynamics tool

Figure 1 illustrates the modelling approach adopted during the study.

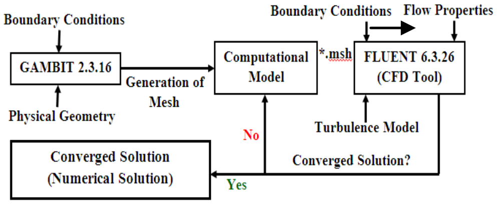

Figure 1: Modelling Approach

The 3-D geometrical model of the wing was created in AutoCAD 2012 and then imported to the mesh generation tool, GAMBIT 2.3.16.

A tetrahedral mesh having 1,285,546 cell volumes was generated on the computational domain using GAMBIT 2.3.16 for facilitating the solution of flow transport equations as shown in figure 2.

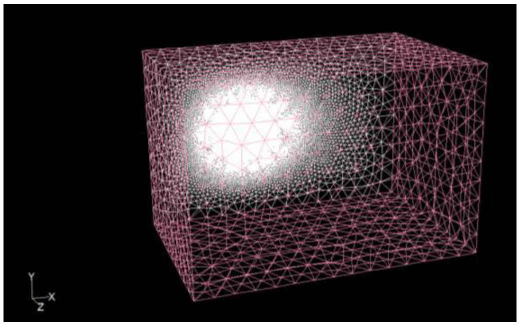

Figure 2: 3-D Mesh created by GAMBIT

The resolution of the mesh was varied in the computational domain for optimizing the accuracy of the solution and to compromise with the computational time as shown in figure 3. On this basis, a finer mesh was established on the regions adjacent to the wing for facilitating the computation of radical changes taking place due to the boundary layer interactions and viscous effects. Volumes of the computational domain, in which the changes related to the flow behaviour are not substantial, were incorporated with a coarse mesh.

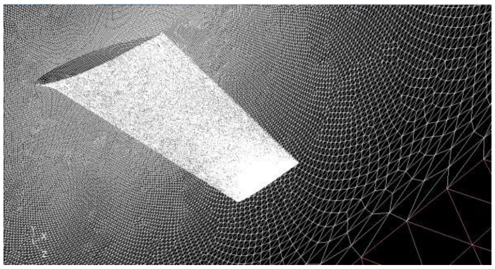

Figure 3: Variation of the Computational Mesh

The computational model was imported to *Ansys Fluent 6.3.26,* the CFD tool used during the analysis. Spalart-Allmaras model was used as the turbulence model since it is widely accepted as a typical model for aerodynamic applications. All related boundary conditions and flow physics were also incorporated in the model. Simulations were run on a 3.2 GHz workstation of 4.0 GB RAM at the Faculty of Engineering of the General Sir John Kotelawala Defence University, Sri Lanka until the solution converged, as shown in figure 4. The process was repeated for all related Mach numbers subjected to 0^0 and 4^0 angles of attack.

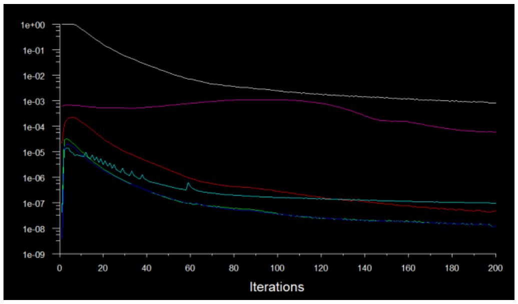

Figure 4: Solution reaching the state of convergence

III. RESULTS AND DISCUSSION

As analogous to the results rendered when modelling the two dimensional tip airfoil, the lift curve is generated as shown Figure 5.

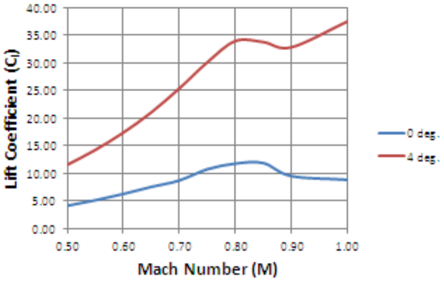

Figure 5: Lift Coefficient vs. Mach Number

It is seen that lift is drastically reduced around M_∞=0.85 with the effects of drag divergence in the transonic regime become more pronounced. The rise in dynamic pressure simultaneously gives rise to drag as shown in figure 6.

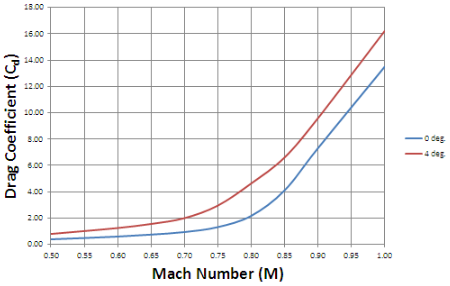

Figure 6: Drag Coefficient vs. Mach Number

The drag divergence for the two dimensional airfoil occurred around M_∞=0.85, whereas it occurs around M_∞=0.8 for the three dimensional wing. The variation of moment shown in figure 7 is mainly due to pressure distribution over the body and qualitatively resembles the variation of the drag coefficient.

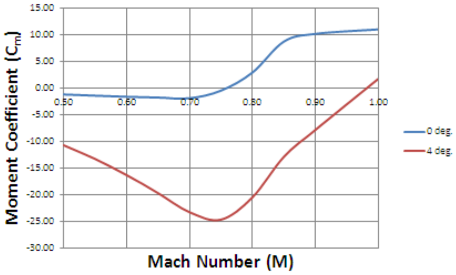

Figure 7: Moment Coefficient vs. Mach Number

The static pressure profile in figure 8 validates the results obtained through intuition and CFD simulation for the infinite two dimensional airfoil in subsonic and transonic regimes as well as the results obtained for a three dimensional wing. Figure 10 shows the velocity vectors throughout the wing and adjacent regions elaborating the variation in velocity and flow separation effects. The magnitude of the velocity vectors retards towards the root and are more prominent at the tip. The pressure coefficient contours are plotted in figure 11.

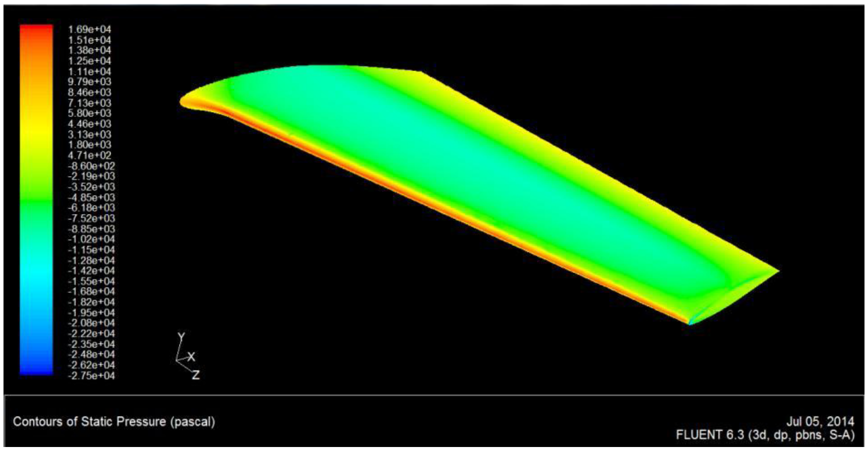

Figure 8: Static Pressure profile

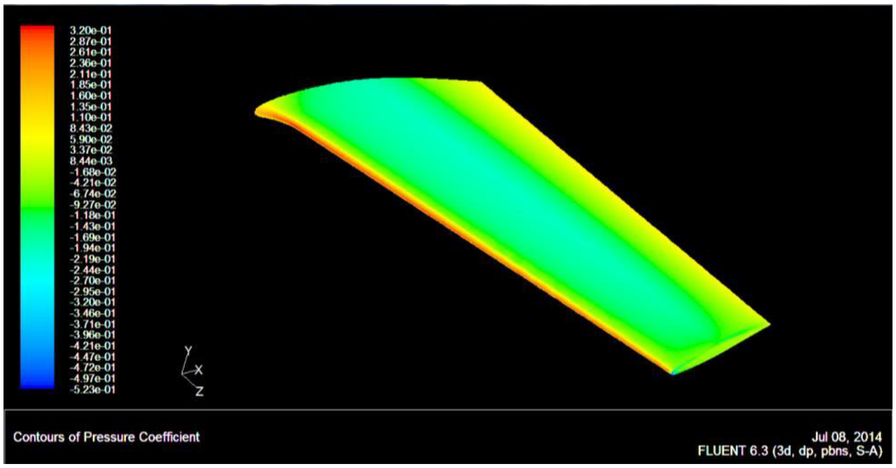

Figure 9: Velocity Vector profile

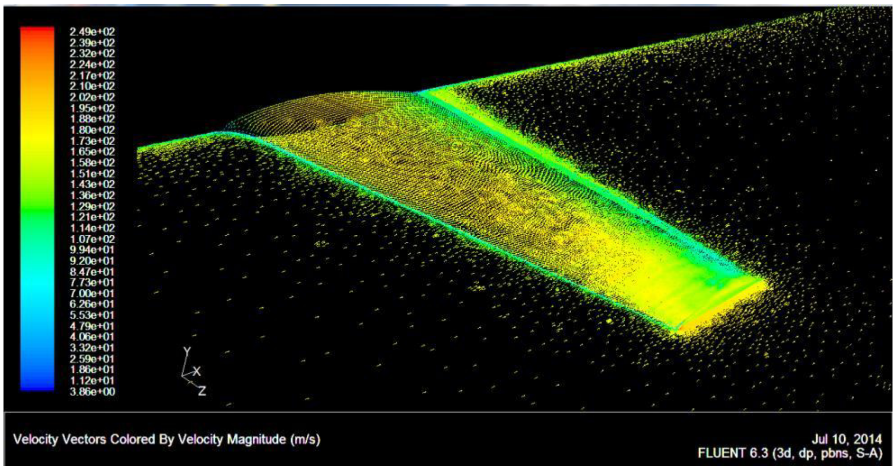

Figure 10: Pressure Coefficient profile

IV. CONCLUSION

A three dimensional analysis of flow physics of a finite wing having a NACA 64A212 airfoil tip and NACA 64A114 root was conducted. The model developed and data generated in this work will serve as the basis for future work in modelling an entire wing body combination for the real aircraft, thereby generating solutions required to conduct a complete performance analysis and possible design modifications to enhance handling qualities of the aircraft.

ACKNOWLEDGMENT

Authors highly acknowledge the support extended by Mr. Ruwan Udayanandana, lecturer (Mechanical Engineering) of the Sri Lanka Institute of Advanced Technological Education in developing the 3-D geometrical model of the aircraft wing.

LIST OF REFERENCES

Anderson, Jr, J.D. (2010), *Aircraft Performance*, McGraw-Hill.

Anderson, Jr, J.D. (2011), *Fundamentals of Aerodynamics*, 4th Edition, McGraw-Hill.

FLUENT 6.3 User's Guide (2006), Fluent Inc.

Tu, J., Yeoh, G.H. and Liu, C. (2008), *Computational Fluid Dynamics – A Practical Approach,* Buttcmurth-Heincmnn publications, UK.

Carmichael, R. (2010), *6A – Series Sections*, Available from http://www.pdas.com/sections6a.html [accessed 10 Feb 2013].

Bandara, R. M. P. S. , & Abeygoonewardene, J. I. (2013). Assessment of Aerodynamic Behaviour of the NACA 64A212 airfoil on the context of Computational Fluid Dynamics. *Proceedings of the KDU International Research Symposium,* pp. 273-277 .

Shang, J. S., & Scherr, S. J. (1986, December). Navier Stokes Solution for a Complete Re-entry Configuration. *J. Aircraft, Vol 23, No 12* , 881-888.

BIOGRAPHY OF AUTHORS

[1]Author is a Senior Lecturer of Aeronautical Engineering of General Sir John Kotelawala Defence University (KDU), Sri Lanka. Her research interests include Flight Dynamics and Cooperative and Autonomous Behaviour of Unmanned Aerial Vehicles. She has served the Sri Lanka Air Force as an Aeronautical Engineer for the past 16 years. Her teaching areas include Aerodynamics, Aircraft Stability & Control and Air breathing Propulsion.

[2]Author is a Senior Lecturer of Mechanical Engineering of General Sir John Kotelawala Defence University (KDU), Sri Lanka. His research interests include Computational Modelling and Optimization of Energy Systems. At present he holds the position of Head of Department of Mechanical Engineering at KDU. He has 16 years of both academic and industrial experience. His teaching areas include Thermodynamics, Heat and Mass Transfer and Computational Fluid Dynamics.