International Journal of Engineering & Technology, 7 (4.13) (2018) 202-204

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper

Aerodynamic performance of X-plane with two different types of inboard stores

Mahadhir A Rahman[1], Mohammad Yazdi Harmin[1*], Mohd Fuad Koslan[2], Mohd Rashdan Saad[3], Mohd Faisal Abdul Hamid[1]

[1]Department of Aerospace Engineering, Faculty of Engineering, Universiti Putra Malaysia, Malaysia
[2]Centre Aerospace Engineering Services Establishment, Malaysia
[3]National Defense University of Malaysia, Malaysia
**Corresponding author E-mail: myazdi@upm.edu.my*

Abstract

This paper presents the investigation of aerodynamic performance of inboard Store-X and Store-Y configurations on the X-plane aircraft model through computational fluid dynamics (CFD) analysis. The X-plane and Store-Y represent the default store and pylon integration while Store-X provides a possibility for other types of store to be integrated. These stores are loosely based upon the two most commonly used by the western and eastern blocks. The resultant lift, drag and moment forces are of interest in order to observe their impact with respect to the two different stores configurations. The finding shows that the aerodynamic impact with respect to Store-X installation on the inboard pylon station is insignificant when compared to default system, hence offers the safety of delivering the Store-X from the X-plane aircraft.

Keywords: *aerodynamic performance; aerodynamic coefficients; computational fluid dynamics.*

1. Introduction

Any modification involving changes of the aircraft shape requires re-evaluation in every aspect of the regulation before the airworthiness status can be issued. This includes installation of additional external role equipment such as camera pod, sensors, tank and radar system. The impact is not limited to the weight and balance, and structural integrity, but also in terms of aerodynamic characteristics and performances. Nowadays, computational fluid dynamics (CFD) analysis has been greatly applied for analyzing the flow characteristics and aerodynamic performances of an aircraft. It has becoming one of the most practical tools, especially during design stage before the development of physical model can be idealized.

Since the computer technologies have been enhanced over the past decade, CFD analysis is able to be employed at much bigger solution domain as well as for solving the governing equations of fluid flow over complex model configuration. CFD is widely accepted as one of the important methods for research and development in the aerodynamic design of an aircraft. For example, CFD simulation of CREATE-AV/Kestrel Solver has been employed for both F-16 and F-22 jet fighters, and the simulation results are well correlated with data from original equipment manufacturer (OEM) in terms of aerodynamic performance characteristics [1]. Moreover, 3D wing of NACA 2412 has been simulated using a well-known CFD software, ANSYS FLUENT and the results have been found to be comparable with the theory of lift generation [2].

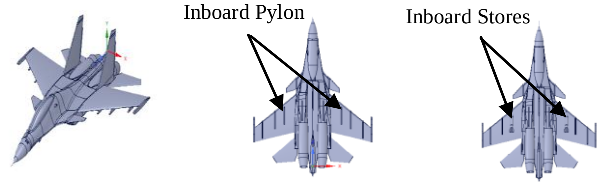
There are many other studies that have been performed to verify the CFD simulation results with the experimental data. These include, among others, the use of 3D RANS CFD software to simulate the canard for SAAB passenger aircraft [3] and blended wing body (BWB) model [4], ANSYS CFX CFD software for different wing designs of a remote control aircraft [5], CATIA and ANSYS FLUENT for Scottish Aviation Bulldog light aircraft [6], general Zonal Euler Solver (ZEUS) to simulate the supersonic missile of SM-2 Block IVA [7] and the use of CFD++ software to study the drag force on a different geometry of nacelle [8]. All these results have been verified with the wind tunnel testing and they are found to be in good agreement. Therefore, the CFD tool has been proven as reliable in computing the aerodynamic performance of the considered model.

In a previous study, the CFD simulation have been investigated on the X-plane model with various size-scale configurations of inboard Store-Y [9]. Simplified CFD scheme through Boolean operation with turbulence model of k-ε has been employed throughout the analyses. The findings have been rather encouraging, with the drawn conclusion of a possibility for another type of stores to be installed at this particular station without affecting the overall aerodynamic performance of the aircraft. This current presented research work is a continuity of that work in [9] with a much better CFD scheme of blockage modelling method and the turbulence model of k-ω has been employed. The k-ω model is selected as it can provide good performance for the computation of the boundary layers in external aerodynamics as well as because it considers subtler interactions between turbulent stress and mean flow when compared with the Reynolds stress model [10].

2. Methodology

In this work, a 1:48 scaled down model of the X-plane aircraft is considered along with the inboard stores of Store-X and Store-Y.

The CAD model of STP format is developed through scanning the physical scaled-down model of the X-plane using the 3D scanner. This enables the model to be imported to a proprietary CAD software and allowing any required repair with respect to gaps, edges and surfaces to be performed. Following this, the assembly procedure has been carried out for integrating the store to inboard pylon of the X-plane model. Finally, they are then defined as single solid body through the merging operation. Figure 1 illustrates the CAD model, showing the isometric view and the location of the inboard pylon and stores. It should be noted that, in this work, only stores that are located at the inboard pylon of the wing are considered for the CFD simulation. In addition, note that the X-plane and Store-Y represent the default store and pylon integration whereas Store-X provides a possibility for other types of store to be integrated.



(a) Isometric view (b) Clean X-plane (c) X-plane with Store

Fig. 1: X-plane CAD model

Following successful modelling of the X-plane and stores models, the corresponding CAD model is then imported to the CFD environment of ANSYS FLUENT. During this transition, there is possibility for the geometry model to lose some of its features due to compatibility issue between these two software. Usually the CAD system fails to meet the criteria with regards to required smoothness and continuity for CFD simulation. As consequences, a repair procedure need to be performed. The repair capability is available within the current CFD environment in order to correct any faulty occurrence in terms of edges, faces and gap. After there is no more faulty remains in the model, the meshing and boundary conditions can then be safely defined.

A CFD scheme of blockage modelling method has been employed instead of Boolean modelling. The Boolean operation subtracts the solid body from the enclosure, hence removing certain degrees of complexity and leaving only the respective fluid domain. This can reduce the accuracy of the CFD simulation. On the other hand, the blockage modelling allows the complexity of aerodynamic shape to be accounted as solid model and therefore it is always remained in the fluid domain. Although the reliability and accuracy of the solution are improved by using blockage modelling, a significant increase in computational time for the convergence of the solution cannot be avoided.

Figure 2 illustrates the computational domain, consisting the fluid surrounding and solid model, and showing its corresponding inlet, outlet and wall as well. It should be noted that the size of the computational domain (or known as an enclosure) is equivalent to the size of the test section 1m × 1m × 2m in width, height and length, respectively. The boundary conditions have been set based on the wind tunnel operational envelope and conditions. Table 1 provides the meshing properties and boundary conditions of the CFD environment. Meanwhile, Figure 3(a) and Figure 3(b) show the example of CFD meshing representation of surrounding wall and surrounding model, respectively.

A mesh convergence analysis has been performed to ensure good approximation of the CFD simulation. This is done by increasing the mesh size of the elements until no more significant improvement in the solutions is found. Figure 4 shows an example of convergence analysis at angle of attack of 15^o for the three considered configurations. The solutions have been normalized by the number of elements of 1.0×10^6. It can be seen that the solutions begin to reach its convergence at 1.6×10^6 elements and beyond this point, the subsequent solutions differ by no more than 0.01%. Once all CFD environment parameters have been satisfactorily defined, the CFD simulation is carried out for a range of velocities and angles of attacks, α.

Table 1: Meshing properties and boundary conditions

Element type	Tetrahedron
No. of element	≈ 1.6×10^6
Size of element	≈ 0.002 m^2
Turbulence model	k-ω
Fluid properties	Air, Sea level

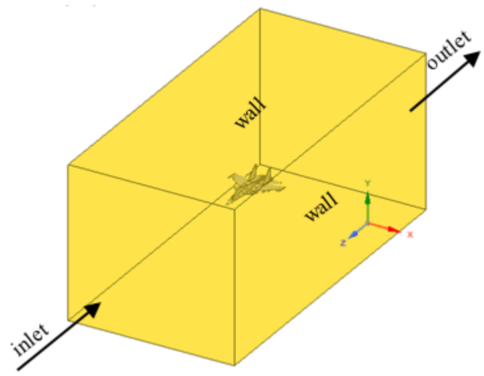
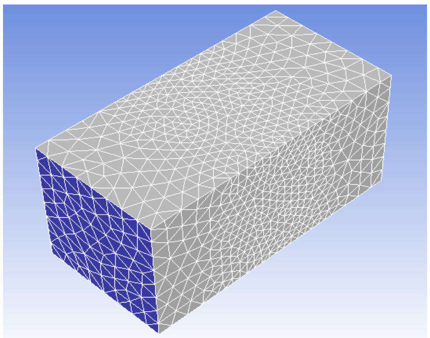
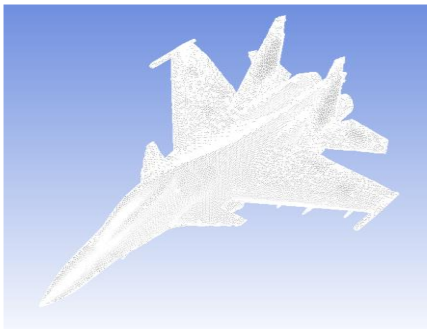


Fig. 2: X-plane model inside the computational domain



(a)



(b)

Fig. 3: Computational domain and X-plane model ready for calculation

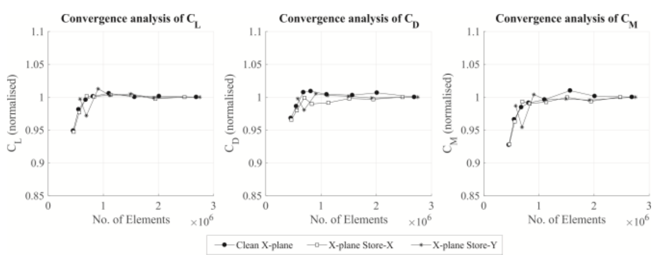


Fig. 4: Convergence analysis of aerodynamic coefficients at α = 15^o

3. Results and discussion

Figure 5 and Figure 6 present the plot trends of lift, drag and moment forces and coefficients, respectively, against a range of angle of attacks (AoA) for three configurations of clean X-plane and X-plane with Store-X and Store-Y. Three subsonic velocities of V1, V2 and V3 are considered with the condition such that V3 > V2 > V1. It should be noted that, due to confidentiality reasons, only the plot trends will be discussed in this section.

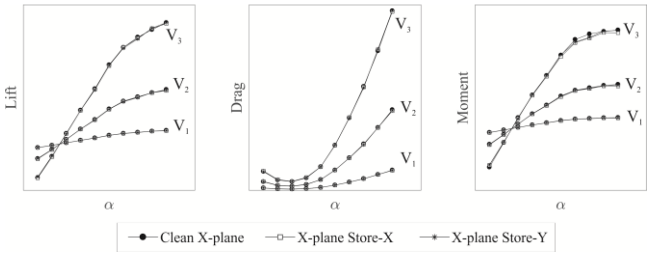


Fig. 5: Lift, drag and moment forces

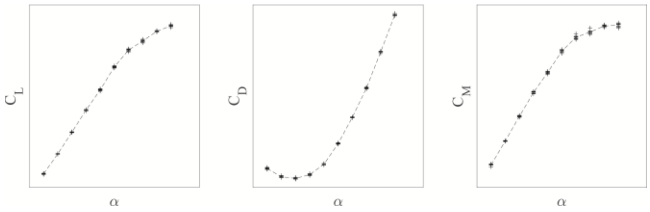


Fig. 6: C_L, C_D and C_M

The lift trends show an increment in magnitude as α increases. It is observed that, at higher α, the lift curve slope of $\delta L/\delta\alpha$ starts to reduce. This indicates that the lift performance of the aircraft has been slightly decreased but nevertheless, no stalling condition has been reached within a range of interest of α. A similar trend can be seen in pitching moment that is the product of lift magnitude and moment arm about the centre of gravity. Meanwhile, the increase in drag magnitude is proportional with the increment in the frontal area and boundary layer thickness as α increases. From the figures, it can be seen that the aerodynamic forces and coefficients are not significantly affected by the installation of Store-X and Store-Y to the inboard pylon location.

Table 2 presents statistical investigation in terms of standard deviation of the aerodynamic coefficients among the clean X-plane and X-plane with Store-X and Store-Y over a range of velocity. This provides the level of dispersion of the corresponding parameter for the range of considered angle of attacks. Even though the standard deviations show an increment as the velocity increases, they are still at a much lower magnitude with the standard deviation of no more than 8% from its mean value. Hence this is again showing that the aerodynamic performance due to the Store-X and Store-Y installation to the inboard pylon station is not significant when compared to the clean X-plane configuration.

4. Conclusion

Two types of store, namely Store-X and Store-Y, are considered at the inboard pylon station with a range of angle of attacks and velocity are taken into account for the CFD simulation. The finding is encouraging, showing that the aerodynamic impact related to lift, drag and moment parameters is insignificant. The standard deviation of aerodynamic coefficients between the three configurations (clean X-plane, X-plane with Store-X and X-plane with Store-Y) is very low. This offers an indication of safe delivery of the Store-X from the X-plane aircraft.

Table 2: Standard deviation for lift, drag and moment

α	Standard Deviation					
	Lift Coefficient		Drag Coefficient		Moment Coefficient	
	C_L	%	C_D	%	C_M	%
α-1	0.005	1.53	0.004	3.95	0.006	2.32
α-2	0.003	2.33	0.004	6.87	0.002	2.36
α-3	0.004	3.60	0.004	7.74	0.004	6.56
α-4	0.006	1.77	0.004	5.38	0.006	2.62
α-5	0.008	1.48	0.003	2.55	0.007	1.89
α-6	0.008	1.02	0.003	1.24	0.009	1.78
α-7	0.011	1.13	0.002	0.47	0.012	1.98
α-8	0.013	1.15	0.003	0.62	0.013	2.01
α-9	0.007	0.54	0.004	0.51	0.006	0.91
α-10	0.012	0.91	0.005	0.56	0.012	1.80

Acknowledgement

The authors acknowledge the support from the Malaysian Ministry of Science, Technology and Innovation (MOSTI) for funding this research work under the ScienceFund Grant RMK-10 with Project No. 04-01-04-SF1243. In addition, the authors are also grateful for the support from Lt. Col. (R) Mohd Ramly Ajir and Dr. Md Amzari Md Zhahir in this work.

References

[1] Dean JP, Clifton JD, Bodkin DJ & Ratclif CJ (2011), High resolution CFD simulations of manoeuvring aircraft using the CREATE-AV/Kestrel Solver. 49th AIAA Aerospace Sciences Meeting

[2] Triet NM, Viet NN & Thang PM (2015), Aerodynamic analysis of aircraft wing. *VNU Journal of Science: Mathematics-Physics* 31, 68-75

[3] Ghoreyshi M, Darragh R, Harrison S, Lofthouse AJ & Hamilton PE (2016), Canard-wing interference effects on the flight characteristics of a transonic passenger aircraft. AIAA Applied Aerodynamics Conference

[4] Waters SM, Vokuijl M, Veldhuis LLM & Gueskens FJJMM (2013), Control allocation performance for blended wing body aircraft and its impact on control surface design. *Aerospace Science and Technology* 29, 18-27

[5] Krishnamurthy S, Jayashankar S, Rao SV, Rochen Krishna TS & Nyamanavar S (2014), CFD analysis of an RC aircraft wing. *International Journal of Mechanical and Production Engineering* 2, 63-8

[6] Lawson NJ, Salmon N, Gautrey JE & Bailey R (2013), Comparison of flight test data with a computational fluid dynamics model of Scottish aviation bulldog aircraft. *Aeronautical Journal* 117(1198), 1273-1291

[7] Frostbutter DA, McGrath BE & Roger RP (2001), Application of computaional fluid dynamics in missile engineering. *Johns Hopkins Apl Technical Digest* 22, 289-301

[8] Trapp LG & Argentieri HG (2010), Evaluation of nacelle drag using computaional fluid dynamics. *J. Aerosp. Technol. Manag.* 2, 145-54

[9] Rahman MA, Harmin MY, Koslan MFS, Saad MR, Romli FI & Ahmad MT (2017), Size scale effects of inboard store-y on x-plane model: a parametric study using CFD. *IOP Conference Series: Materials Science and Engineering* 270, 012006

[10] Versteeg HK & Malalasekera W (2007), *An introduction to computational fluid dynamics: the finite volume method*. Pearson Education.