IOP Conf. Series: Materials Science and Engineering **270** (2017) 012006 doi:10.1088/1757-899X/270/1/012006

Size-scale effects of inboard Store-Y on X-plane model: A parametric study using CFD

M A Rahman[1,2], M Y Harmin[1,*], M F S Koslan[2], M R Saad[3], F I Romli[1] and M T Ahmad[3]

[1]Department of Aerospace Engineering, Faculty of Engineering, Universiti Putra Malaysia, Serdang, Selangor, Malaysia
[2]Center of Aerospace Engineering Services Establishment, Royal Malaysian Airforce, Subang Air Base, Malaysia
[3]Department of Mechanical Engineering, Faculty of Engineering, National Defence University of Malaysia, Sungai Besi, Malaysia

*myazdi@upm.edu.my

Abstract. This paper presents the computational fluid dynamics (CFD) study on the effects of the Store-Y integration to the inboard pylon of X-plane model to its aerodynamic performance. Parametric study in terms of the size-scale of the Store-Y model is simulated at various angles of attack and air velocities. The resultant lift, drag and moment force parameters are recorded in each simulation and then analysed. The findings show that the size-scale effect of Store-Y model is insignificant to the aerodynamic performance, hence offers a possibility of other types of store to be integrated at this station without affecting the overall aircraft performance.

1. Introduction

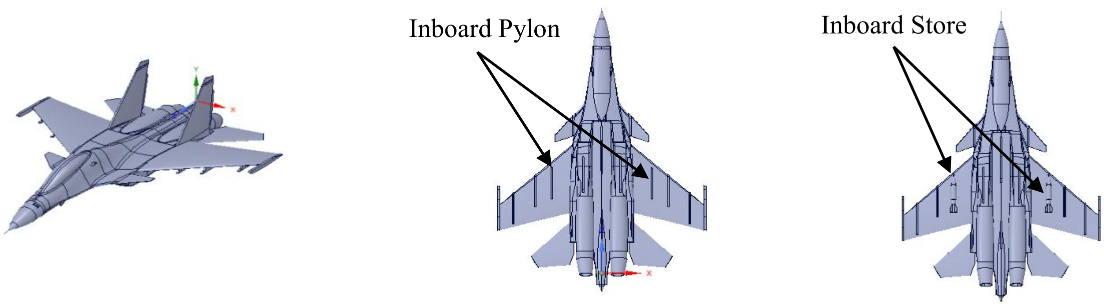
Computational fluid dynamics (CFD) has been widely used today for analyzing the characteristics of the flow that passes by an object. The development in CFD simulation is always closely tied with the enhancement in computer technology, which enables the governing complex fluid dynamic equation of Navier-Stoke to be solved in a much larger solution domain [1]. Because of this capability, CFD is becoming one of the most practical tools in aerospace engineering for the computation of the aircraft's aerodynamic performance. For instance, the CFD simulation of CREATE-AV/Kestrel Solver has been used for both F-16 and F-22, and the results are found to be in very good correlation with the original equipment manufacturer (OEM) data in terms of aerodynamic performance characteristics [2]. On the other hand, the 3D wing of NACA 2412 has also been simulated using a well-known CFD software of ANSYS FLUENT and the results are comparable with the theory of lift generation [3].

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 [4], ANSYS CFX CFD software for different wing design of remote control aircraft [5], General Zonal Euler Solver (ZEUS) to simulate the supersonic missile of SM-2 Block IVA [6] and the use of CFD++ software to study the drag force on a different geometry of nacelle [7]. 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.

One of the main challenges in CFD simulation is to acquire an effective meshing while assuring the accuracy of the results [8]. As the complexity of the model increases, the overall meshing procedure is also becoming more complicated, leading in increment of number of elements as well as resulting to a much higher computational time. In addition, the CFD solution may become stiffer to be solved and, in the worst case scenario, the numerical integrator might even fail to converge. Having this in mind, an effective strategy for the setup of CFD environment is required to achieve computational efficiency of the CFD simulation. In this work, the CFD simulation of ANSYS FLUENT is used to compute the aerodynamic performance of the X-plane wind tunnel model. The CFD environment is set up based on the wind tunnel operational envelope and conditions, along with the use of standard k-ε model as the turbulence model.

2. Simulation Modelling

A scaled down model of 1:50 ratio of the X-plane aircraft is considered in this study. The dimension of the model is 0.3m in length, 0.11m in span and 0.45m in height. This studied model represents the X-plane wind tunnel model in a wind tunnel test section of 1m × 1m × 2m in width, height and length, respectively. Initially, the physical scaled down model of the X-plane is scanned using the 3D scanner in order to produce a Computer Aided Design (CAD) model that is compatible with CAD environment of SOLIDWORK. Following this modelling process, the assembly procedure is done to integrate the Store-Y to the inboard pylon of the X-plane. They are then merged together, forming a single solid body of the model. Figure 1 shows CAD models of the clean X-plane and the X-plane with the stores. Note that, only the stores that are located at the inboard pylon of the wing are considered in this study.



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

Figure 1: X-plane CAD model

The CAD model is then imported to the CFD environment of ANSYS FLUENT, where meshing is done and boundary conditions are defined. Note that, the size of the computational domain (or known as an enclosure) is equivalent to the size of the test section and the boundary conditions have been set based upon the wind tunnel operational envelope and conditions. During the transition from the CAD to the CFD environment, compatibility issues might arise that can cause the model to loss some of its features. Under those circumstances, the repair procedure needs to be carried out to identify any faulty occurrence in terms of edges, faces and gap in the model. Once all necessary repairs have been made, the Boolean operation is then employed for subtracting the solid body from the enclosure, leaving only the respective fluid domain. This operation enables simplification of the meshing process, especially involving the complex region.

The meshing procedure begins by considering a coarse mesh density setting with the employment of the tetrahedron mesh. To ensure a good approximation of the solution, a mesh convergence analysis is carried out by increasing the mesh density of the fluid domain until the solutions have satisfactorily reached its convergence. If the solution is unable to acquire its convergence, the minimum element size, curvature size and skewness parameters have to be redefined accordingly. Figure 2 provides a sample of the mesh convergence analysis for the X-plane with the Store-Y, where the solutions are normalized with the corresponding solutions of mesh density of 1×10^6 elements. It can be seen that the solutions

IOP Conf. Series: Materials Science and Engineering **270** (2017) 012006 doi:10.1088/1757-899X/270/1/012006

begin to reach its convergence at mesh density of 6×10^5 elements and it is noted that beyond this point, the solutions only differ by no more than 0.02%. Furthermore, Figure 3(a) and 3(b) provide the CFD representation of the surrounding wall and the surrounding model meshing of the considered study, respectively.

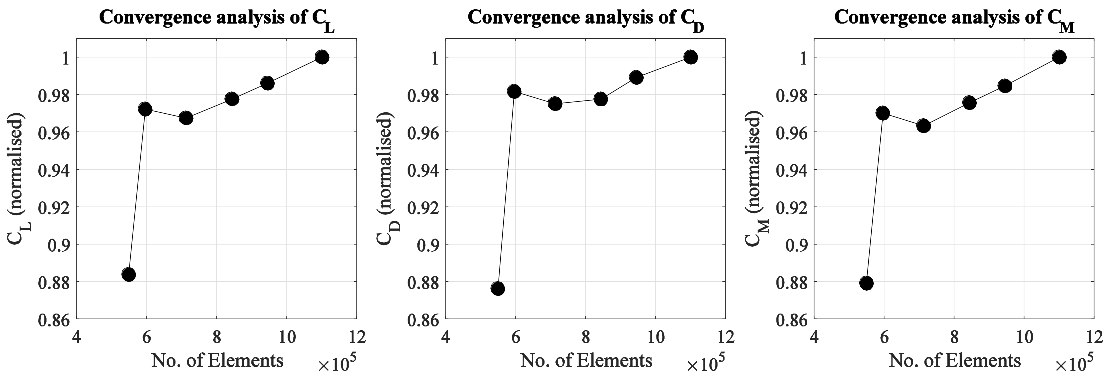
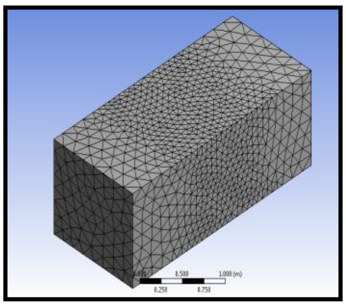
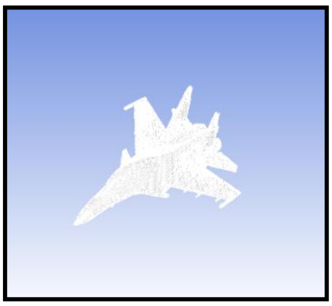


Figure 2: Mesh convergence analysis



(a) Surrounding wall meshing



(b) Surrounding model meshing

Figure 3: X-plane in CFD environment

Once all CFD environment parameters have been satisfactorily defined, the X-plane model is then simulated. A parametric study in terms of store dimensions is carried out under the selected range of velocities and angles of attack, α. A turbulence model of k-ε is employed as it can offer the simplest level of closure, resulting into relatively low computational time [9]. The fluid solver of sweeping and semi-direct methods are utilized to iterate the governing fluid equations to the specified convergence criterion [10].

3. Results and Discussion

Due to the confidentiality of publishing the technical data to the general public, only the plot trends are provided in this paper. The results are presented in terms of lift, drag and moment against independent variable of angle of attack, α with corresponding velocities of namely V1, V2 and V3. The study is intended to provide initial assessment on the effect of the store size to the aerodynamic performance.

Figure 4 shows a comparison in aerodynamic performance between the clean X-plane and X-plane with Store-Y. In terms of the lift trend, the increment in lift magnitude with increasing α is due to the simulation conditions that are below the stall region where the lift curve slope, $\delta L/\delta\alpha$ is positive value. Note that, at a much higher α, the lift begins to lose its performance with of a soft stall characteristic. Meanwhile, in terms of the drag trend, an increase in drag magnitude is mainly due to the increment of the frontal area and boundary layer thickness. Lastly, the negative slope of moment trend indicates the pitching magnitude of nose up condition. Overall, the aerodynamic performance of the X-plane with Store-Y is slightly underperformed as compared to the clean X-plane. This is mainly due to the flow over the wing being disturbed by the stores, resulting in increase of boundary layer separation. Since the store is located near to the fuselage, very minimal degradation in aerodynamic performance should be expected.

On the other hand, Figure 5 provides a comparison of the aerodynamic performance of the X-plane for three different size-scales of the Store-Y with scaling ratio of 0.75:1, 1:1 and 1.25:1. Note that the aerodynamic impact due to the store size is still very minimal and hence, the previous discussion is also applicable here as well. Table 1 presents the standard deviation of the simulation results, which describes the level of dispersion among the parametric results of the store models for each considered α and velocity. It can be observed that the standard deviations of the aerodynamic performance are very small, with a deviation of at most 1.01 in magnitude. This shows an insignificant impact of store sizes for the store located at the inboard pylon to the aerodynamic performance.

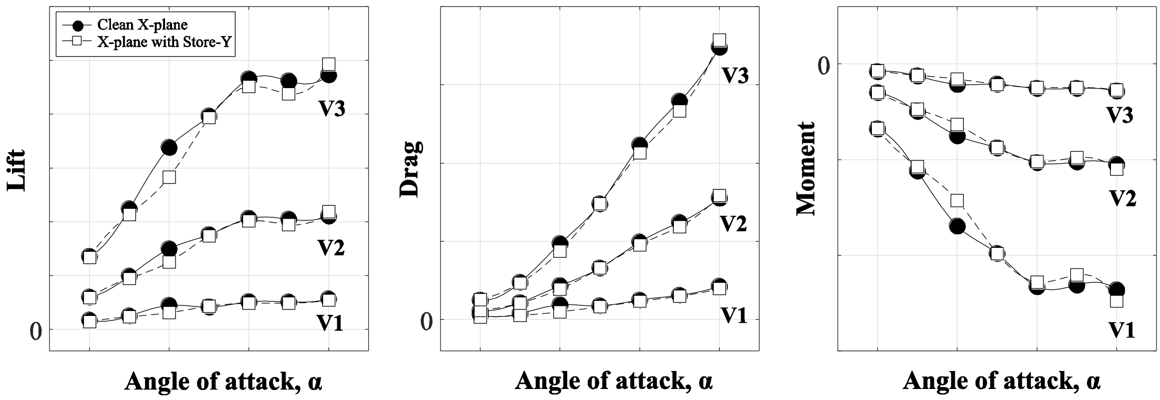


Figure 4: Aerodynamic performance of clean X-plane and X-plane with Store-Y

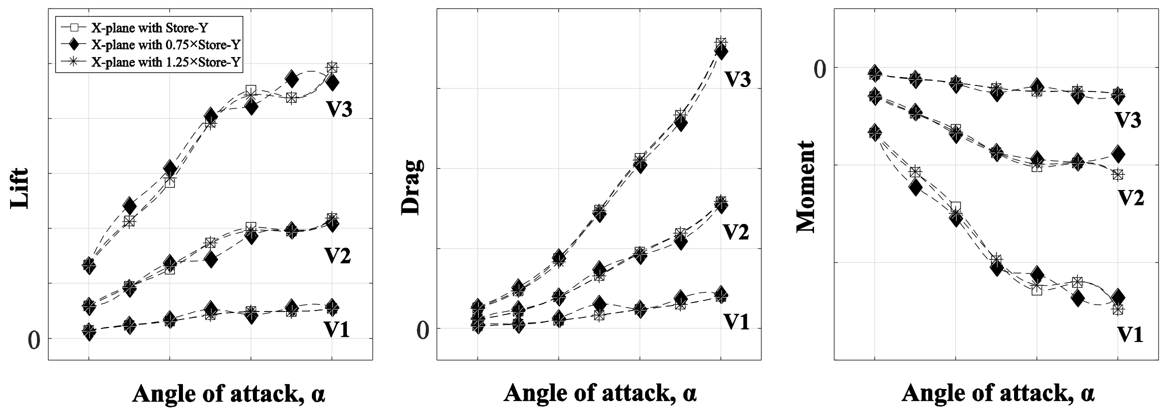


Figure 5: Aerodynamic performance of X-plane with various size scale of Store-Y

4. Conclusion

The X-plane wind tunnel model with different size-scale of Store-Y is modeled and simulated on the CFD simulator. The finding is encouraging, where the size-scale effects of the inboard Store-Y on the X-plane model is found to be insignificant. This provides a possibility for another type of stores to be installed at this particular station without affecting the overall aerodynamic performance of the aircraft. Further investigation should be concentrated in validating the findings of this study through wind tunnel testing.

Table 1: Standard deviation of the simulation results on the size-scale effects of Store-Y

α	Standard Deviation								
	Lift			**Drag**			**Moment**		
	V1	**V2**	**V3**	**V1**	**V2**	**V3**	**V1**	**V2**	**V3**
α-1	0.06	0.01	0.02	0.04	0.02	0.04	0.05	0.02	0.04
α-2	0.03	0.09	0.84	0.00	0.07	0.12	0.02	0.08	0.36
α-3	0.10	0.29	0.68	0.08	0.01	0.14	0.13	0.25	0.57
α-4	0.29	0.83	0.36	0.40	0.23	0.10	0.11	0.02	0.18
α-5	0.17	0.36	0.73	0.02	0.10	0.19	0.13	0.21	0.45
α-6	0.21	0.07	1.01	0.22	0.27	0.26	0.09	0.13	0.40
α-7	0.07	0.25	0.74	0.08	0.11	0.29	0.04	0.51	0.35

Acknowledgement
The authors acknowledge the support for this work from Malaysian Ministry of Science, Technology and Innovation through the research grant funding under ScienceFund RMK-10: Project No. 04-01-04-SF1243. The authors are also grateful for the support given in this study by Lt. Col. (R) Mohd Ramly Ajir and Dr. Md Amzari Md Zhahir.

References

[1] Tannhill J C, Anderson D A and Pletcher R H 1997 *Computational Fluid Mechanics and Heat Transfer* CRC Press
[2] Dean J P, Clifton J D, Bodkin D J and Ratclif C J 2011 *AIAA Aerospace Sciences Meeting*
[3] Triet N M, Viet N N and Thang P M 2015 *VNU Journal of Science: Mathematics-Physics* **31** 68-75
[4] Ghoreyshi M, Darragh R, Harrison S, Lofthouse A J and Hamilton P E 2016 *AIAA Applied Aerodynamics Conference*
[5] Krishnamurthy S, Jayashankar S, Rao S V, Rochen Krishna T S and Nyamanavar S 2014 *International Journal of Mechanical and Production Engineering* **2** 63-8
[6] Frostbutter D A, McGrath B E and Roger R P 2001 *Johns Hopkins Apl Technical Digest* **22** 289-301
[7] Trapp L G and Argentieri H G 2010 *J. Aerosp. Technol. Manag.* **2** 145-54
[8] Koslan M F S, Ahmad Zaidi A M, Othman M Z, Abdullah S and Tanakodi S 2013 *Modern Applied Science* **7** 23-8
[9] Tu J, Yeoh G H and Liu C 2012 *Computational Fluid Dynamics* Elsevier
[10] Kohnke P 1994 *ANSYS Theory Reference Release 5.6*