Computational Fluid Dynamic Simulation (CFD) and Experimental Study on Wing-external Store Aerodynamic Interference of a Subsonic Fighter Aircraft

Tholudin Mat Lazim, Shabudin Mat, Huong Yu Saint

The main objective of the present work is to study the effect of an external store to a subsonic fighter aircraft. Generally most modern fighter aircraft is designed with an external store installation. In this project a subsonic fighter aircraft model has been manufactured using a computer numerical control machine for the purpose of studying the effect of the external store aerodynamic interference on the flow around the aircraft wing. A computational fluid dynamic (CFD) and wind tunnel testing experiments have been carried out to ensure the aerodynamic characteristic of the model then certified the aircraft will not facing any difficulties in stability and controllability. In the CFD experiment, commercial CFD code is used to simulate the interference and aerodynamic characteristics of the model. Subsequently, the model together with an external store was tested in a low speed wind tunnel with test section sized 0.45 m × 0.45 m. Result in the two-dimensional pressure distribution obtained by both experiments are comparable. There is only 12% deviation in pressure distribution found in wind tunnel testing compared to the result predicted by the CFD. The result shows that the effect of the external storage is only significant at the lower surface of the wing and almost negligible at the upper surface of the wing. Aerodynamic interference is due to the external storage were mostly evidence on a lower surface of the wing and almost negligible on the upper surface at low angle of attack. In addition, the area of influence on the wing surface by store interference increased as the airspeed increase.

Keywords: computational fluid dynamic (CFD), wind tunnel testing, validation, aerodynamic interference.

1 Introduction

Fighter aircraft are mostly designed to carry stores such as launcher or external tank under the wing. When these stores are installed, the flow on its surrounding components such as the control surfaces can be considerably changed. This may introduce several aerodynamic interference characteristic such as changes in aerodynamics force, increase in turbulence and possibly flow separation, These phenomena may introduce adverse effect on other aircraft components such as the horizontal tail and vertical stabilizer and consequently may affect the controllability and stability of the aircraft.

Research on external store installation is complex and extensive. It covers several research areas such as aerodynamic, structure, flutter, physical integration, trajectory prediction, aircraft performance, stability analysis and several multiple engineering disciplines. However, the focus of this work is to study the aerodynamic interference particularly on the change in aerodynamic characteristics. The aerodynamic characteristics are prerequisite for the other analysis, since the aerodynamic data are required for a subsequent aircraft structural analysis, stability analysis, performance analysis and store trajectory analysis. The investigation of aerodynamic characteristics in the external store clearance program usually involved complex flow field study with multi component interferences. Usually flow of such nature is investigated through wind tunnel testing besides the empirical methods.

The main objective in this study is to identify the interference effect of a subsonic fighter aircraft that is currently used by Royal Malaysian Air force with the present of external store installation. A generic model of one of the subsonic fighter aircraft used by Royal Malaysian Air Force was chosen for the study. Wind tunnel testing and computational fluid dynamics (CFD) simulation were conducted to investigate these interference effects. The low speed wind tunnel with size 450 mm × 450 mm was used to conduct the experiments and commercial CFD software was used for the simulation. Other milestones in this study include the verification and validation process and the suitability of applying a commercial CFD code for predicting the wing and external store aerodynamics interference effects.

2 Simulation and experimental works

The methodology adopted to conduct the study consists of a few steps. The first and foremost is to obtain digitized wing section geometry. The digitization process was done using Photomodeller software. Then the second step is to construct a scale model of the wing based on the digitized wing geometry using Numerical Control Machine (CNC). Then several series of experiment were carried out upon the scale model in the wind tunnel at a low speed which approximately 22.8 m/s. The digitized wing geometry was also used in the CFD simulation. Gambit preprocessor software was used to produce the necessary mesh. The setup was then simulated using Fluent 5 CFD software and the CFD simulation was carried out with various physical models, numerical algorithms, discretization method and boundary conditions. At the final step, the study was

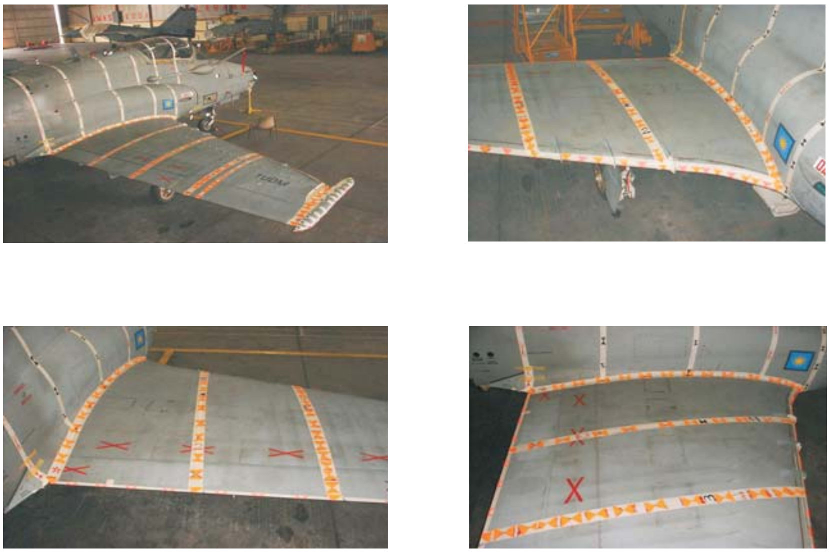

Fig. 1: Photographs of the marked wing surface at the various projection

wrapped up by comparing both the computed and measured results in investigating further the nature of the interference effect of a wing and external storage configuration.

3 Aircraft wing external geometry digitization

Digitization of the wing geometry is vital in order to obtain an adequate aircraft model geometry that represents the real aircraft. A Photomodeler 3.0 software is used to capture and digitize the aircraft wing external geometry. The software captures the image of 84 photographs taken at various angles of the aircraft wing. These photos were taken using a digital camera. A number of points have been marked on the aircraft wing and the adjacent fuselage part using the masking tape as shown in Fig. 1. The size of the markers was designed in such a way that the size will be visible clearly and sharp in the photographs taken at certain distance away. This was determined using the relationship between the number of pixels and the distance from the camera. The placement and location of the markers are determined based on the profile of the wing. The high curvature area was placed with denser markers. This figure also shown a total of 84 photographs were used to generate the wing profile and some part of the fuselage.

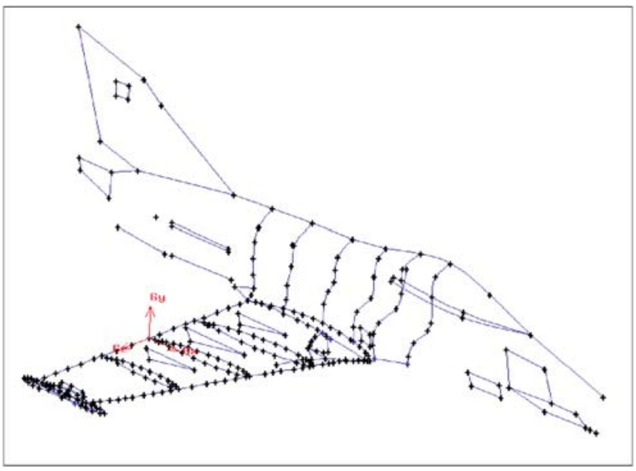

Fig. 2: Aircraft geometry produced by Photomodeller

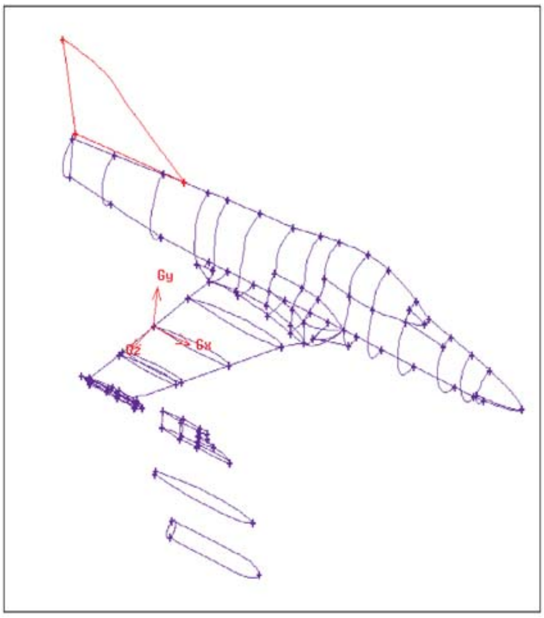

Fig. 3: Digitized wing geometry after smoothed with CAD

Output from the digitization process is a set of coordinates conforming to the wing geometry as shown in Fig. 2. Majority of the coordinates were on the wing surface and wingtip pylon. Unfortunately, the wing geometry image is less quality in term of accuracy and perfection. Therefore, CAD software is used to smooth the image. After made minor adjustment, the image becomes as in Fig. 3.

3.1 Wind tunnel testing

A wing model is required to perform the wind tunnel testing. Therefore, a 20 % scale wing model of a fighter aircraft has been fabricated by using CNC machine. The model was made from a single solid piece of an Aluminum-

-Alloy that having nine conduits each on the upper and lower surface. Fig. 4 show the semi-span model of a generic fighter aircraft taken from the digitized geometry produced by Photomodeller.

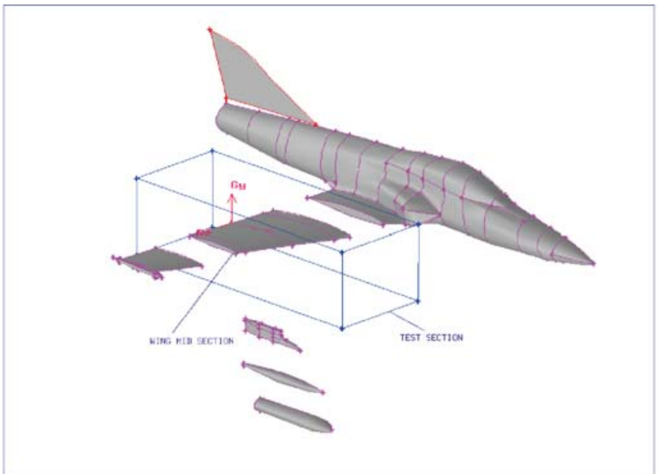

Fig. 4: Semi span model of a generic fighter aircraft

The figure also indicates 3 main part of the wing which includes a root section, mid section and tip section. But the external storage is installed at the mid section. Then we only decided to fabricate the mid section. Furthermore, we unable to test the full set of the wing model due to the limitation of the size of the wind tunnel test section. The model has three main stations for pressure measurement study located at the chord wise that parallel to each other with equal distance placing. Every station was stationed with static pressure-taping point on upper and lower surface respectively. Besides built the mid wing model, we also built a 1/5 scale model of launcher and pylon as the external stores. These external storages were design in such a way that they are easily secured and removed from the wing section. Fig. 5 shows the complete assembly of this aircraft wing together with the external storage, inside the test section. The experiments have been conducted using two different configurations of the wing model. The first configuration is without the external storages. Meanwhile, the second configuration is with the external storages. Both configurations were tested at zero angle of attack at two different speeds, which are 22 m/s and 27 m/s.

3.2 Computational fluid dynamic simulation

In the CFD simulation, the mid wing was simulated at two conditions. In the first condition, the mid wing have been mesh into 111 239 elements. Meanwhile, and the second condition it have been mesh into 221 112 elements as shown in Fig. 6a and 6b. The flow was simulated at the speed 22 m/s, incompressible flow and at laminar consideration. Fig. 6c shows the simulation for wing with external storage with 122 158 elements.

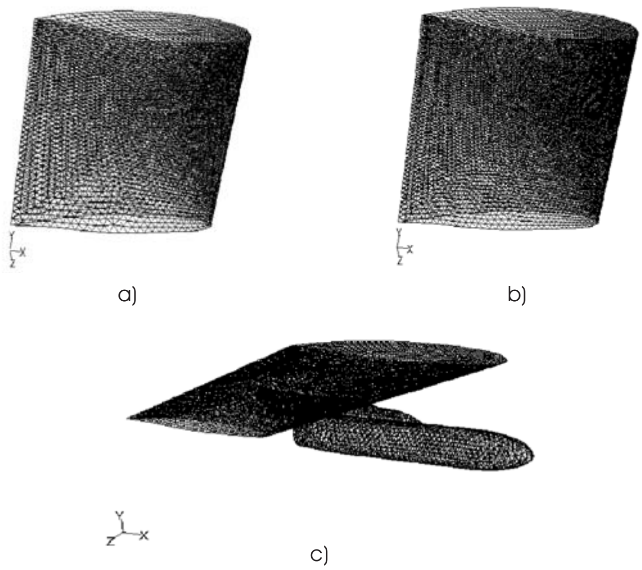

Fig. 6: CFD model surface meshes:
a) mesh for wing in tunnel, 111 239 elements, b) mesh for wing in tunnel, 221 112 elements, c) mesh for wing and store in tunnel, 122 158 elements

Fig. 5: Model installation inside the wind tunnel

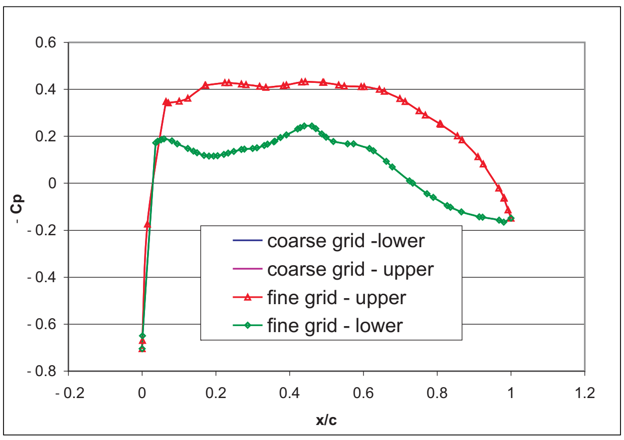

Fig. 7: Pressure distributions at the mid span for upper and lower surface

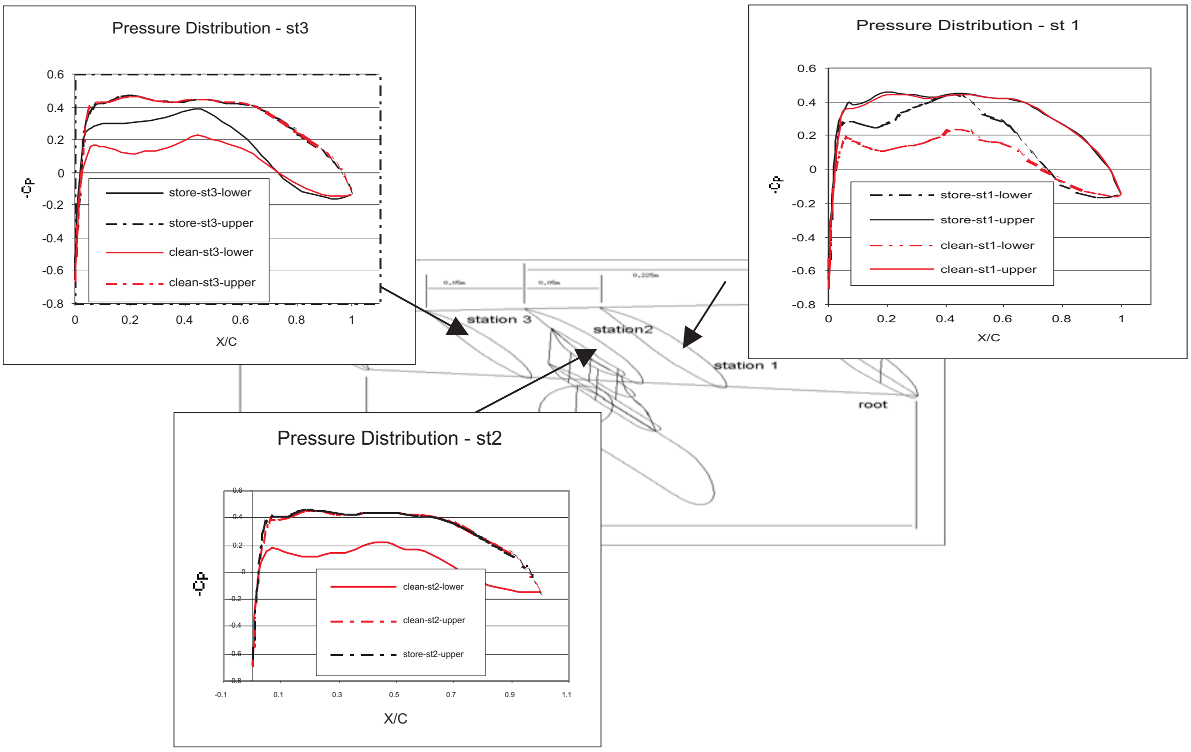

Fig. 8: Pressure distribution for a various chord wise location

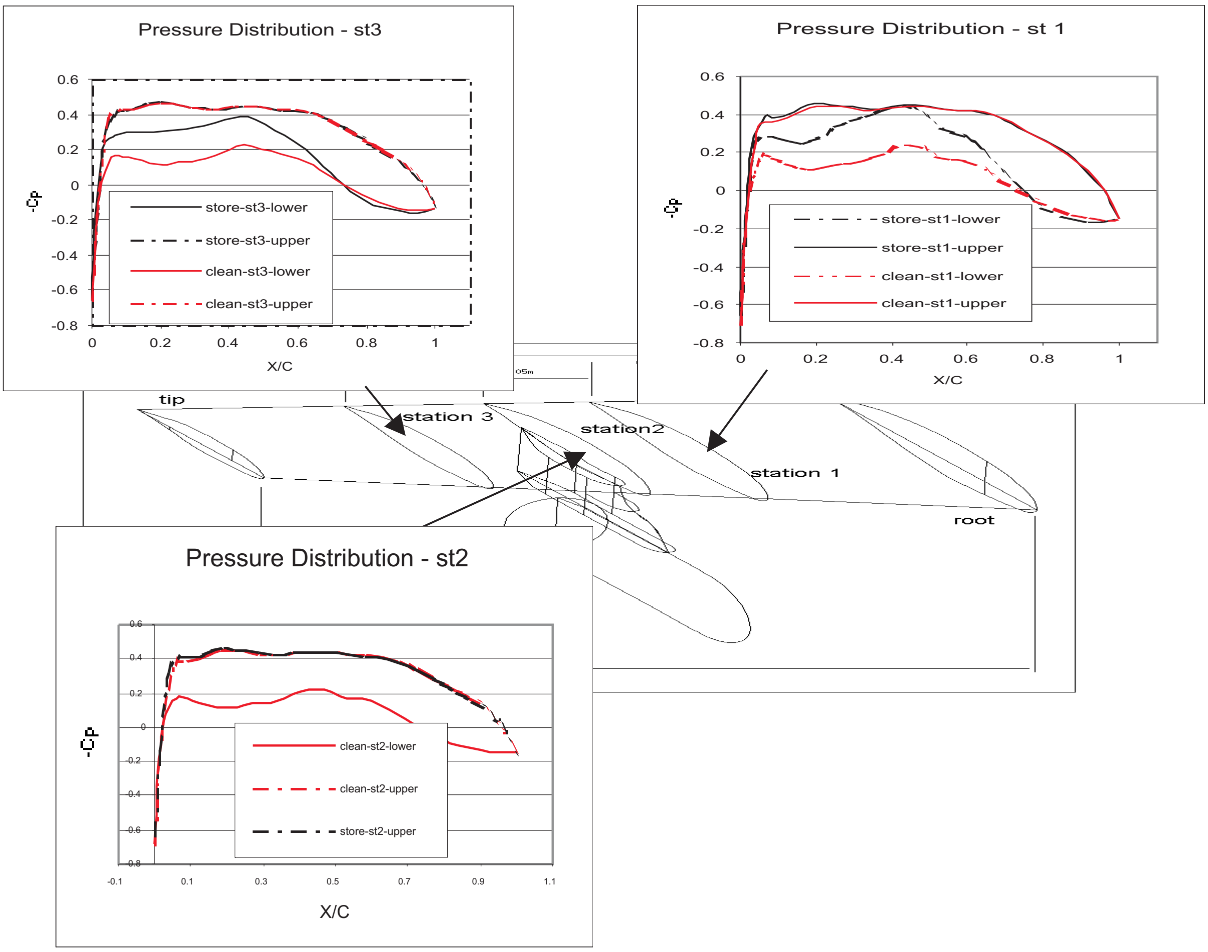

Fig. 9: Pressure coefficient at three different span wise locations

4 Results

4.1 Wind tunnel testing results

After conducted a series of experiment, the pressure distribution at mid span for the upper and lower surfaces was plotted as shown in Fig. 8.

From these results we found that at station 1, the difference in pressure coefficient is only 3 % on the upper surface of the wing due to the external store installations. The pressure coefficient shows there are substantial differences by having external configuration at the lower surface. Station 2 and 3 indicate the same phenomenon that there is a little difference in pressure distribution on the upper surface. The lower surface shows some reduction in pressure distribution compared to the upper surface. These experimental results give an initial indication that the flow at the upper surface will not be severely affected by the external storage configuration compared to the lower surface.

4.2 Computational fluid dynamics results

Fig. 9 shows the results of computational fluid dynamic simulation for the upper and lower surfaces of this aircraft. At station 3, it is found that the coefficient of pressure is almost constant and same from the leading edges to trailing edge and the value is not much different by having the external installation at the upper surface. This shows that the external storage has not affected the flow at the upper surface. In contrast the pressure coefficient is much differed at a lower surface by having the external configuration compared to the clean wing configuration. This shows that the external storage is only affecting the lower surface. The same phenomenon also happen at station 1 where the coefficient of pressure has not changed at the upper surface but there are some reduction about 12 % in coefficient of in the lower surface. The simulation at station 2 also provides the same results.

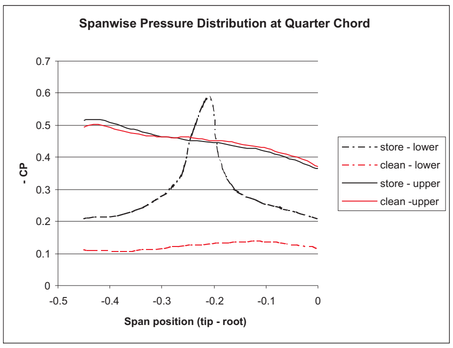

Fig. 10: External store interference on pressure distribution

5 Analysis and discussion

5.1 Comparison between CFD and experimental works

The study shows that the value of pressure coefficient on the upper surface predicted by simulation is around 0.4 compared to 0.6 performed by the experimental study. There is about 12 % difference. In the experimental study, problem during the setup of experiment such as misalignment in determining the angle of attack, accuracy of the model, blockage effects and wind tunnel calibration can significantly influenced the result. Even though the wing is machined accurately by the computer numerical control (CNC), there is still doubt on the accuracy of the model. Moreover, the fluid level of the manometer used for measuring pressure was fluctuating between always from 1 to 2 cm.

5.2 Discussion

In the study we observed that the external store configuration only affects the lower surface of the wing. Fig. 10 shows the pressure distribution at the quarter chord point along the span wise location from the tip to the root. At the upper surface, the pressure distribution is almost constant from the span wise position, which is from tip to the root of the wing. The pressure coefficient at the lower surface was reduced by 40 % compared to the upper surface. With the external storage, the pressure distribution at the lower surface was increased around 20 % compared with the clean lower surface, but there was sudden increased in pressure distribution at the position −0.2 span wise location where the external storage was mounted to the wing.

Conclusion

The experimental study at speed 22.8 m/s and computational fluids dynamic simulation has been performed on the wing and store configuration in this project. The results show the that the flow over the upper surface of the wing has not affected much when the pylon and launcher are installed. The study also shows that the flow over the lower surface is much affected by the presence of external storage. The static pressure around the wing is about 12 % higher than the simulated values.

References

[1] Manoj K. Bhardwaj, Rakesh K. Kapania, Reichenbach E., Guru P. Guruswamy: "Computational Fluid Dynamics/Computational Structural Dynamics Interaction Methodology For Aircraft Wings". *AIAA Journal*, Vol. **36** (1998), No. 12, p. 2179–2185.

[2] Tomaro Robert F., Witzeman F. C., Strang W. Z.: "Simulation Of Store Separation For The F/A-18c Using Cobalt". *Journal of Aircraft*, Vol. **37** (2000), No. 3, p. 361–367.

[3] Prewitt N. C., Belk D. M., Maple R. C.: "Multiple-Body Trajectory Calculations Using the Beggar Code". *Journal of Aircraft*, Vol. **36** (1999), No. 5, p. 802–808.

[4] Brock J. M., Jr, Jolly B. A.: *Application of Computational Fluid Dynamics at Eglin Air Force Base*. 1988, SAE 985500.

[5] Shanker V., Malmuth N.: "Computational And Simplified Analytical Treatment Of Transonic Wing/Fuselage/Pylon/Store Interaction". *Journal of Aircraft*, Vol. **18** (1981), No. 8, p. 631–637.

[6] Hirsch: "Numerical Computation of Internal and External Flows". Volume 1. Brussels: Wiley, 1988.

[7] Jameson A.: "Re-Engineering the Design Process through Computation". *Journal of Aircraft*, Vol. **36** (1999), No. 1.

[8] Spradley L. W., Lohner R., Chung T. J.: ”Generalized Meshing Environment for Computational Mechanics”. *AIAA Journal*, Vol. **36** (1996), No. 9, p. 1735–1737.

[9] Mavriplis D. J.: “Viscous Flow Analysis Using A Parallel Unstructured Multigrid Solver”. *AIAA Journal*, Vol. **38** (2000), No. 11, p. 2067–2075.

[10] Kallinderis Y., Khawaja A., Mcmorris H.: “Hybrid Prismatic/Tetrahedral Grid Generation for Viscous Flows around Complex Geometries”. *AIAA Journal*, Vol. **34** (1996), No. 2, p. 291–298.

[11] Koomullil R. P., Soni B. K.: ”Flow Simulation Using Generalized Static and Dynamic Grids”. *AIAA Journal*, Vol. **37** (1999), No. 12, p. 1551–1557.

Tholudin Mat Lazim, Ph.D
e-mail: tholudin@fkm.utm.my

Shabudin Mat, M.Sc
e-mail: shabudin@fkm.utm.my

Department Aeronautics & Automotive
Faculty Mechanical Engineering
Universiti Teknologi Malaysia
81310, UTM Skudai
Johor Malaysia

Huong Yu Saint, M.Eng

Royal Malaysian Airforce
Wisma Pertahanan
Jalan Padang Tembak
50634, Kuala Lumpur

