

EXPERIMENTAL AND COMPUTATIONAL FLUID DYNAMIC (CFD) STUDIES ON HALF-MODEL TRANSPORT AIRCRAFT

Iskandar Shah Bin Ishak, Shabudin Bin Mat, Tholudin Bin Mat Lazim and Mohd Zailani Bin Awang

Department of Aeronautics and Automotive,
Faculty of Mechanical Engineering,
Universiti Teknologi Malaysia,
81310 Skudai, Johor,
MALAYSIA

ABSTRACT

This paper aims to highlight the wind tunnel testing techniques and the Computational Fluid Dynamic (CFD) studies on a half-model aircraft configuration. A 9% scale generic model of 130-seaters transport aircraft with a 1.028m semi-span is used for the experimental works. The experiment is conducted at Universiti Teknologi Malaysia – Low Speed Tunnel (UTM-LST) which is currently the largest tunnel of its kind in Malaysia with a test section of 2m (breadth) X 1.5m (height) X 5.8m (length) and the maximum wind speed of 80 m/s. The experimental is carried out with a Reynolds No. of 1.3×10^6 . In the CFD work, a commercial CFD code, Fluent 5.3 is used to simulate the aerodynamic characteristics of this subsonic transport aircraft. It is found that the result obtained from the wind tunnel testing is agreeable with the results simulated by the CFD. The experimental result also indicates that the curve of pitching moment coefficient versus angle of attack has a negative slope which depicts the aircraft is statically longitudinal stable.

Keywords: Half model, wind tunnel testing, computational fluid dynamic (CFD).

1. INTRODUCTION

The implementation of wind tunnel testing and CFD is a crucial stage in the analysis process. For this paper, a 9% scale generic model of 130-seaters transport aircraft is experimented at UTM-LST. CFD is used then on the model. Both the results will be discussed by the rest of this paper.

2. WIND TUNNEL TESTING

2.1 UTM -LST

The experiment is conducted at Universiti Teknologi Malaysia – Low Speed Tunnel (UTM-LST). The Reynolds No. of the testing is 1.3×10^6 , with a freestream speed of 60 ms^{-1} . A half-model balance is used as a measurement tool.

Table 1: Correction Factors & Moment Transfers

Symbol	Meaning	Dimension
δ	Wall Correction	0.115
$\epsilon sb_{\text{wing}}$	Wing Solid Blockage Correction	0.00012
$\epsilon sb_{\text{fuselage}}$	Fuselage Solid Blockage Correction	0.00945
ϵ_{wake}	Wake Blockage Correction	0.02163
ΔX Fore/Aft	Moment Transfer	0.11 m
ΔY	Moment Transfer	0.04 m
ΔZ	Moment Transfer	0 m

2.2 A Half-Model of Generic Transport Aircraft

A 9% scale generic model of 130-seaters transport aircraft with a 1.028m semi-span is used for the experimental works.

Table 2: Specifications of the Model

Symbol	Meaning	Dimension
S	Wing Area	0.252 m^2
\bar{c}	Mean Aerodynamic Chord	0.339 m
$b/2$	Half-Span	1.028 m
V_{wing}	Wing Volume	0.00072 m^3
V_{fuselage}	Fuselage Volume	0.058 m^3

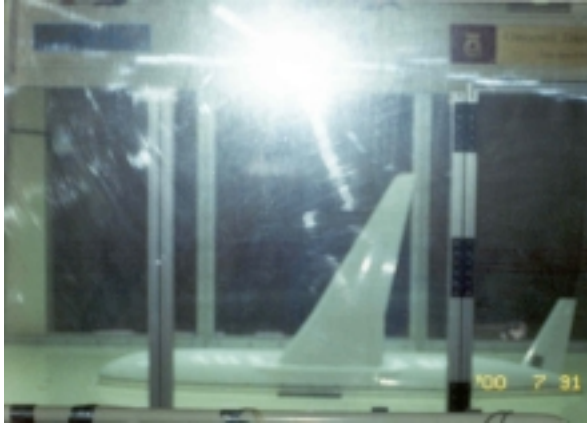


Fig 1: Wind tunnel testing

2.3 Layout of Data Reduction

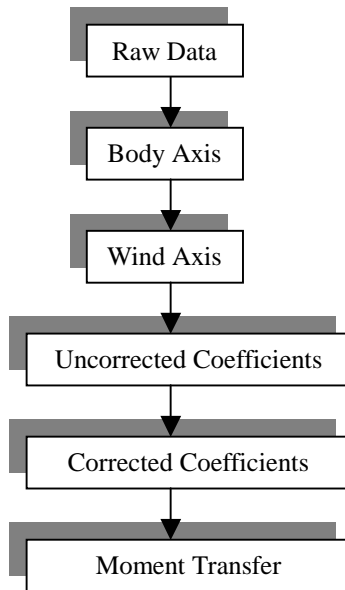


Fig 2: Flow chart of data reduction process

3. COMPUTATIONAL FLUID DYNAMIC (CFD)

The model is then being simulated in Computational Fluid Dynamic (CFD) by using Fluent 5.3 processor. The simulation study carried out in a computational domain with a dimension of 5.50 meter tunnel length, 1.35 meter tunnel width and 1.75 meter tunnel height. The element generated for this simulation is tetrahedral/hybrids mesh. Over 300 000 cells are generated using CFD Pre Processor.

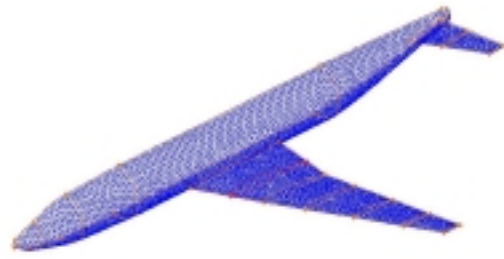


Fig 3: Surface mesh for the whole model using triangular mesh

The model is simulated at a Mach number of 0.17 (or equivalent to the velocity of 60 m/s) with variations of angle of attack, $\alpha = 0^\circ, 4^\circ, 8^\circ$ and 10° respectively. The Reynolds No. is 1.3×10^6 . Throughout the rest of the case study, the ideal gas condition is assumed. Free stream turbulence kinetic energy and its dissipation are presumed to be 5% of mean flow. The standard wall function is applied on the wing and the fuselage.

After the convergence condition is achieved, the number of cells can be improved by using solution-adaptive refinement, that mean adding cells where they are needed in the mesh, thus enabling the features of the flow field to be better resolved.

4. RESULTS

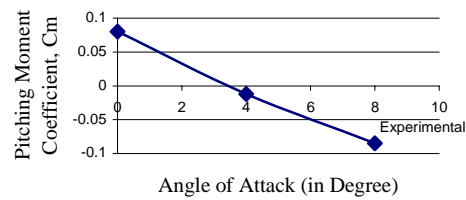


Fig 4: Pitching Moment Coefficient (C_m) vs. Angle of Attack (α)

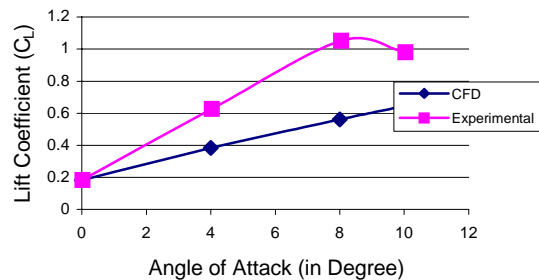


Fig 5: Lift Coefficient (C_l) vs. Angle of Attack (α)

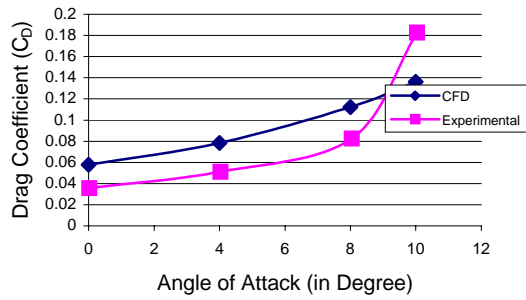


Fig 6: Drag Coefficient (C_D) vs. Angle of Attack (α)

5. CFD GRAPHIC RESULTS

5.1 At angle of attack, α = 0°

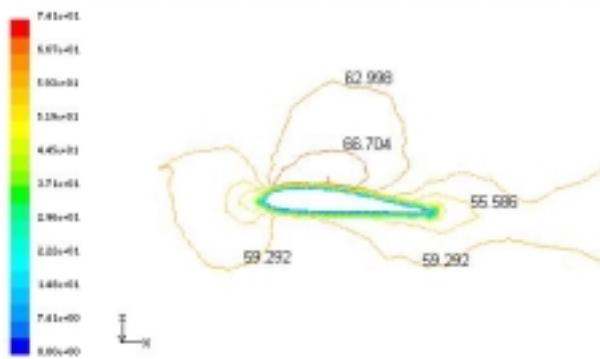


Fig 7: Contour of Velocity in m/s at Station 1

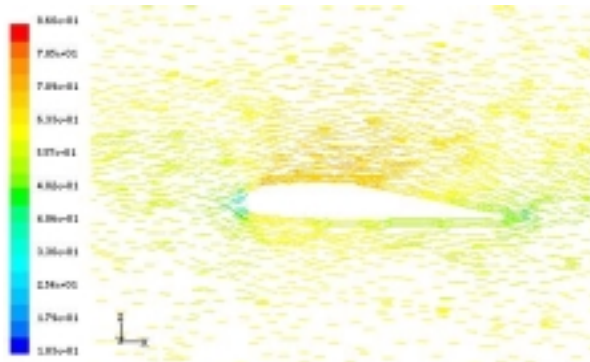


Fig 8: Velocity vector in m/s Station 1

5.2 At angle of attack, α = 10°

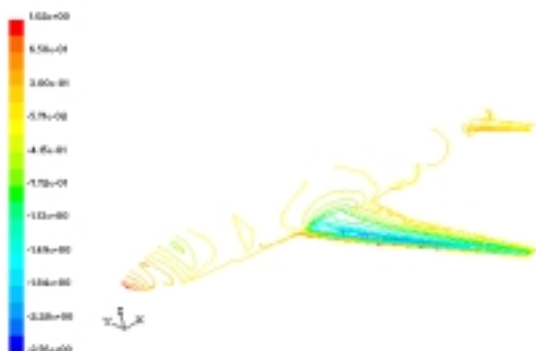


Fig 9: Contour of Pressure Coefficient (C_p)

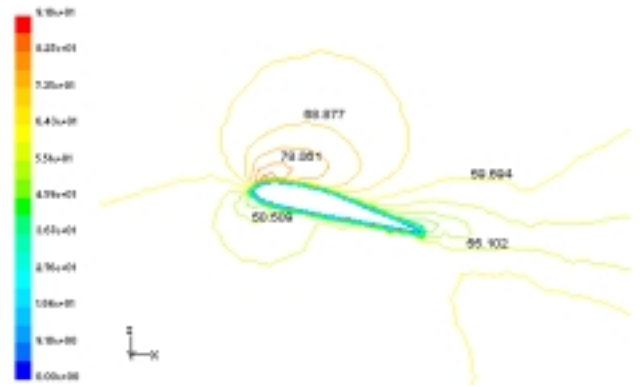


Fig 10: Contour of velocity in m/s Station 1

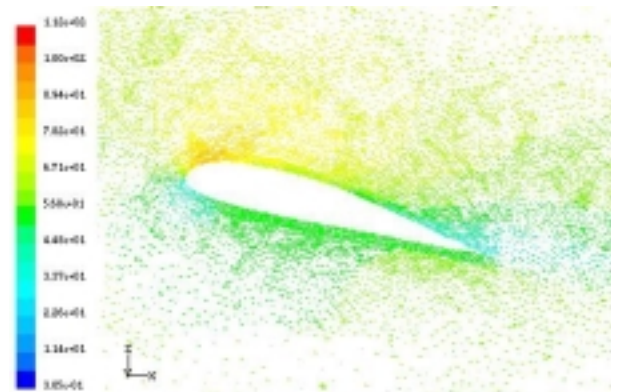


Fig 11: Velocity vector in m/s Station 1

Note: Station 1 is a cross-section at 0.09m from the centerline of the fuselage in spanwise direction

6. DISCUSSION

Fig. 4 indicates that that the aircraft is a statically longitudinal stable as it has met the two criteria for longitudinal static stability, which are :

- ◆ $C_{m_0} > 0$
- ◆ $C_{m_\alpha} < 0$

It also can be said that that the testing and the data-reduction techniques are *good* as the testing result **satisfying** with the actual aircraft characteristics.

Fig. 5 depicts the coefficient of lift, C_L found for both experiments. Experimental study gives the maximum coefficient of lift to be 1.14 at the stalling angle 9° degree. Coefficient of lift is found increased with the increasing of angle of attack. Nevertheless the stall angle is hard to be found through CFD. This both results show that the wind tunnel results agreed well with the CFD at the low angle of attack.

Fig. 6 shows the comparison for drag coefficient for both experiments. Experimental and simulation study shows that the drag is increased when angle of attack increases especially at a low angle of attack. The

comparison has been made and it is found that there are about 15 % difference between the 2 methods but the curve of these two studies shown an agreeable slope. Coefficient of drag is found to be 0.095 at the stalling angle, from the experimental result.

7. RECOMMENDATIONS

In the study we observed that the experimental study and CFD is a good tools to determine the aerodynamic forces of the aircraft model. Nevertheless there are some differences when the coefficient of lift and drag compared in this study. This is may be caused by inaccuracy of the CFD model that simulated compared to the experimental model. In the future, the real image of the model will be taken through the digitize software called Photo-Modeller Pro 3.0. Furthermore, no correction has been applied in this CFD Simulation which un doubt will mislead the result.

8. CONCLUSION

The experimental study at the wind speed of 60 m/s and computational fluids dynamic simulation has been performed on the half model of generic transport aircraft in this project. The results show that the coefficient of lift and drag are well agreed at low angle of attack even there are 15 percent differences. These 2 methods can be used to determine the aerodynamic forces in the early stage of designing the aircraft.

9. REFERENCES

1. Dimitri J. Mavriplis (2000). "Viscous Flow Analysis Using A Parallel Unstructured Multigrid Solver". AIAA Journal, Vol. 38, No 11, 2067-2075.
2. Yannis Kallinderis, Aly Khawaja and Harlen McMorris (1996). "Hybrid Prismatic/Tetrahedral Grid Generation for Viscous Flows around Complex Geometries". AIAA Journal, Vol 34, No 2. 291-298.
3. R.P. Koomullil and B.K. Soni (1999). "Flow Simulation Using Generalized Static and Dynamic Grids". AIAA Journal, Vol. 37, No 12, 1551-1557.
4. Barlow, J.B. (1999) " Low-Speed Wind Tunnel Testing ", 3rd Edition, John Wiley & Sons, Inc., USA.
5. Richard J. D. Poole, AMR Inc. Canada, Wind Tunnel Testing at UTM-LST, Feb 2002