Journal of Aeronautical
-Science and Engineering-, Vol.2

May 20, 2015

Computational Fluid Dynamic on Double Delta Wing

Khairul Adzha B Abd Halim,[a], and Shabudin B Mat,[a]

[a)]Department of Aeronatics, Automotive and Ocean Engineering, Universiti Teknologi Malaysia, Malaysia

Corresponding author:[a]shabudin@fkm.utm.my

Paper History

Received: 18-Feb-2015
Received in revised form: 26-Apr-2015
Accepted: 19-May-2015

ABSTRACT

Basic concept of aircraft wing design is based on airfoil section. Time flies, evolution in aircraft wing design shows the desire of mankind to improve the speed, agility, maneuverability of an aircraft. It is proven that aircraft with delta or double delta wing design fulfill this desire. Basic concept of delta wing is triangular planform of the wing that same with the Greek symbol (Δ) and double delta wing is delta wing with a 'kink' or leading edge extension. This research aims is to obtain aerodynamic characteristic (C_l, C_d) of double delta wing using computational fluid dynamic and compare the result with wind tunnel experiment. In this work, the geometry of the double delta wing used was constructed using 2 x 10^6unstructed mesh elements. Turbulence model that been used in this research is k-ω turbulence model. The simulation was run at Reynolds Number of 1 x 10^6 and 2 x 10^6 and with variation of pitch angle from 0^0 to 20^0.

KEY WORDS: *Computational Fluid Dynamic; Double Delta Wing.*

NOMENCLATURE

C_l	Coefficient of Lift
C_d	Coefficient of Drag
Re	Reynolds Number
α	Angle of Attack
k	Turbulent Kinetic Energy
ω	Specific Dissipation Rate
ε	Turbulent Dissipation

1.0 INTRODUCTION

The conventional aircraft wing design concept is basically based on airfoil section which comes with low drag at cruising speed. The evolution of aircraft shows that desire of mankind to have a high maneuverability and agility of an aircraft wing design. This desire demands considerable improvements in the aerodynamic characteristics and its related control. To fulfil that desire, many researches had been developed and one of the researches is based on delta wing design. It is best to understand the aerodynamic characteristics from the model it first. Those aerodynamic characteristic can be gain in numbers of ways such as flight test, drop test, water tunnel and even computational fluid dynamic. In this particular study, those aerodynamic characteristic will be obtain by Computational Fluid Dynamic (CFD) simulation. CFD data have potential to be reliable data if a correct turbulent model, geometry, and boundary condition is used. Modelthat will be using in this study is double delta wing with (65^0/25^0) sweep angle.

2.0 PAPER FORMAT

2.1 Computational Fluid Dynamic (CFD)

Computational Fluid Dynamics or CFD can be described as the use of computers to solve the governing equations for fluid flow in any given situations [1]. CFD represent sets of data for given flow configurations at different Mach number, Reynolds number, etc. same as like wind tunnel. Unlike wind tunnel that heavy, costly, unwieldy device, CFD is much more preferable nowadays because it can be carrying around and can be accessed remotely by people on terminals that can be thousands of miles away from the computer itself.

In order to modelling fluid flow for various geometries, Fluent software is the most preferable software. According to John D Anderson,Fluent supported 2D triangular/ quadrilateral, 3D tetrahedral/ hexahedral/ pyramid meshes and it is also refining or coarsening grid based on the flow solution [2]. There are several turbulent viscous models in Fluent that is appropriate with this thesis such as Spalart-Allmaras model, Standard k-□ model and etc. There are few considerations to choice the turbulence model such as the physics encompassed in the flow, level of accuracy required, the available computational resources and the amount of time available for the simulation.

From the several turbulent viscous model, *k-ω* turbulence model was chosen because from the previous study show that this model can predict the flow separation process with higher accuracy [3]. Wilcox (1998) developed *k-ω* model to give better compute low Reynolds number effects, compressibility, and shear flow spreading. It is an empirical based model with transport equations for turbulence kinetic energy (k) and specific dissipation rate (ω). Transport equations used in Fluent for Wilcox's model are as follows.

$$\frac{\delta}{\delta t}(pk)+\frac{\delta}{\delta x_i}(pku_i)=\frac{\delta}{\delta x_i}\left[\left(\mu+\frac{\mu_i}{\sigma_k}\right)\frac{\delta k}{\delta x_j}\right]+G_k-Y_k+S_k \qquad (1)$$

$$\frac{\delta}{\delta t}(p\omega)+\frac{\delta}{\delta x_i}(p\omega u_i)=\frac{\delta}{\delta x_i}\left[\left(\mu+\frac{\mu_i}{\sigma_\omega}\right)\frac{\delta \omega}{\delta x_j}\right]+G_\omega-Y_\omega+S_\omega \qquad (2)$$

where:

G_k - generation of turbulent kinetic energy arises due to mean velocity gradient
G_m - generation of rate, w
Y_k - dissipation of kinetic energy, k
Y_w - dissipation rate, w
α_k, α_w - turbulent Prandtl numbers

2.1 Double Delta Wing

Double delta wing is essentially a delta wing with a "kink" in its leading edges that forms the shoulder where the leading edges of the strake (or Leading Edge Extension, LEX) and main wing intersect [4]. A delta wing is a wing shape when viewed from top like a Greek symbol (Δ) forms like a triangle. It sweeps sharply back from the fuselage with the angle between the leading edge of the wing often high as 60 degrees and the angle between the fuselage and the trailing edge of the wing mostly around 90 degrees.

Aerodynamic investigation of flow over delta wing configurations have been performed for many years. Typical fact that known by previous research are the flow separates already at low angles of attack at the highly swept leading edges [5]. The flow over a delta wing is a vortex dominated flow field. The vortex formed attached to the upper surface of the wing. The flow can be describes as a movement of a part of the flow from the lower to the upper surface into spiral type of motion. This can be seen in Figure 2.1. Verhagen, Jenkins, Kern, and Washburn[6] in their study show that when $\alpha < 10^o$, the two vortices remained separated and hardly interacted. Beyond this angle of attack, the interaction between the two vortices became more pronounced. This was believed to indicate that the breakdown of the strake vortex was causing the wing vortex to burst.

As mention by Lu Zhi-Yong[7] in his study, there are two types of vortex breakdown have been recognized which is in bubble form and the other is the spiral form. Bubble form of breakdown occurs because rapid expansion of the core forming a bubble-like structure that is nearly axisymmetric while for a spiral form, the vortex centerline deforms into a spiral without any appreciable growth in core size. Figure 2.2 shows the spiral and bubble form of vortex breakdown.

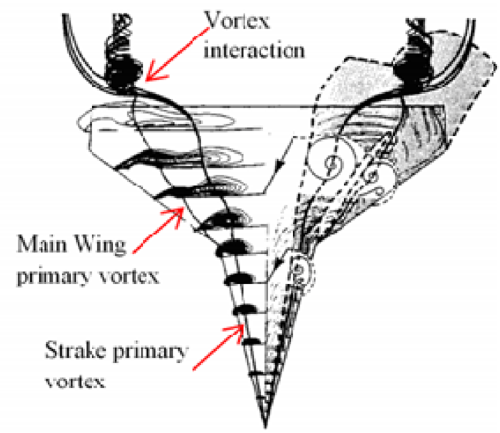

Figure 2.1: Vortex Flow around Double Delta Wing (courtesy from Numerical investigation of high incidence flow over a double-delta wing. Journal of Aircraft, Vol. No 32, 1995).

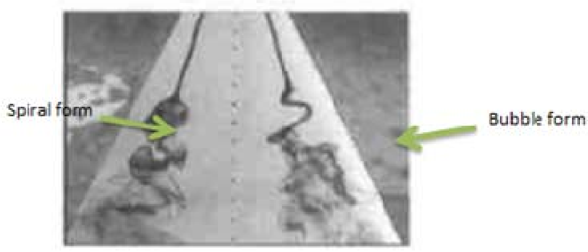

Figure 2.2: Form of vortex breakdown (courtesy from Study on Forms of Vortex Breakdown over Delta Wing. Chinese Journal of Aeronautics, Vol. 17 No 1, 2004).

3.0 METHODOLOGY

In this chapter, all those sequences during the progress of this thesis will be briefly explained. The idea was divided the whole process into two parts according to the two semesters of study. In this thesis the aerodynamic characteristics obtained by using computer fluid simulation. However, the result from this thesis will be compared with wind tunnel experiment results.

3.1 Flow Chart

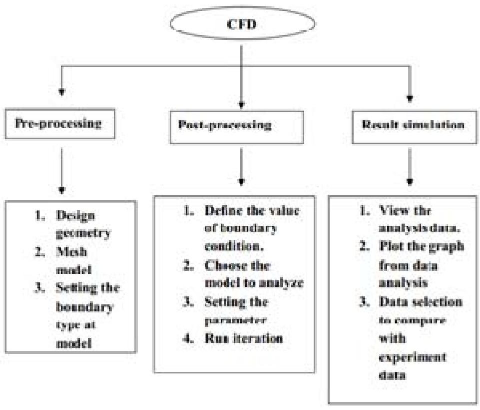

Figure 3.1: Process of CFD simulation

3.2 Pre-Processing

Pre-processing is where to prepare the input data before a simulation is run. At first, by using others third party software like SolidWorks to set up the geometry. The solid drawing of this particular double delta wing was given in Figure 3.2. Then, this model was subtracted from a block of 5800 mm x 2000 mm x 1500 mm which is similar to the control volume in testing section of UTM Low Speed Tunnel. The solid area will be representing as the fluid moving around the model.Block with cavity of the double delta wing then will be cut to be half since we only consider pitch angle in this experiment. There is not consideration of yaw angle in this experiment. The cavity will be half body of double delta wing.

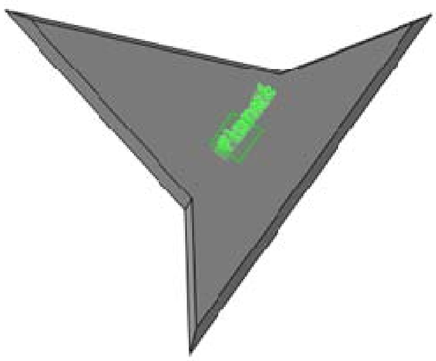

Figure 3.2: Double Delta Wing Model

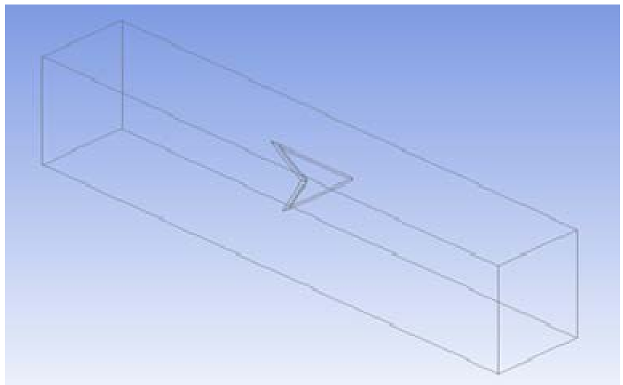

Figure 3.3: Half Body of Double Delta Wing

The solid cavity block then meshed to direct the flow around the model. The solid must be converted first as a paraSolid before being import to ANSYS Workbench. The best grid was chosen in order to get the best result. Too much fine grid will contribute to a longer time as the computer need more time to compute the flow field.

3.3 Post Processing

This is where value of boundary condition was defined and iteration calculation started. In this study, FLUENT will be used as the solver and 3d type of analysis was chosen.

Table 3.1: Boundary Condition

Re	$1x10^6$		$2x10^6$	
(α^o)	x-component (m/s)	y-component (m/s)	x-component (m/s)	y-component (m/s)
0	30.51	0	61.02	0
5	30.394	2.659	60.788	5.318
10	30.046	5.298	60.093	10.596
15	29.470	7.897	58.941	15.793
20	28.670	10.435	57.340	20.870

3.4 Result Simulation

All the results can be show either by contour of pressure, density, velocity and many more. The results also can be save in txt file to do further analysis and comparison for other result. CFD simulation and wind tunnel measurement were compared associated with the difference of all forces and the moment. All data will be tabulated and graph of data analysis between CFD simulation and Wind Tunnel test will be plot.

4.0 RESULT

4.1 Introduction

This part shows the result that had been gained in from the computational fluid dynamics. The main concern for this study is to get the flow visualization and determined the aerodynamic characteristics of the double delta wing. Comparison between CFD result with wind tunnel testing focused on aerodynamic characteristic which is lift coefficient and drag coefficient.

4.2 Flow Visualisation

Figure 4.1 shows the velocity contour of double delta wing at 20% of chord. Range for contour velocity is made to be fixed from 0 to 42 m/s to make comparison from various angles of attack. As we can see from the contour of velocity of the model, it is shown that as the angle of attack increases, the vortex formed by the model is become more severe.

Figure 4.2 show the contour of total velocity of double delta wing at 70% of chord while Figure 4.3 shows the contour of total pressure of double delta wing at 70% of chord. Basic of fundamental fluid dynamic that say when velocity is higher at certain place, the pressure at the place will be the lowest and vice versa. This phenomenon can be seen from the Figure 4.2 and Figure 4.3.

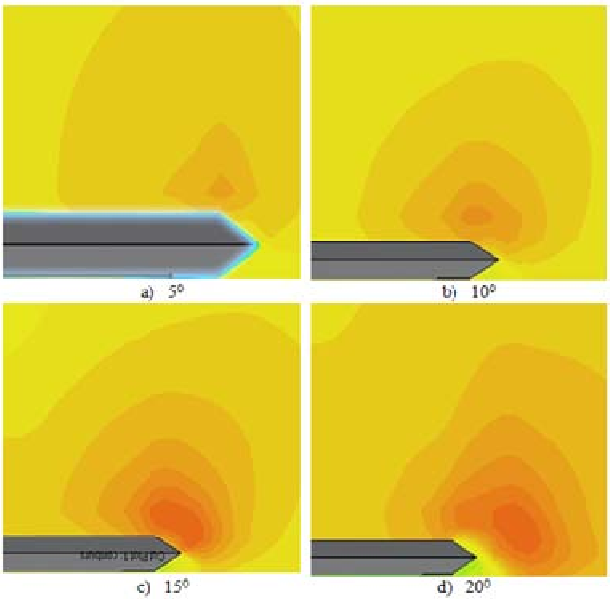

Figure 4.1: Front view contour of total velocity at 20% chord at Re = 1 x 10^6

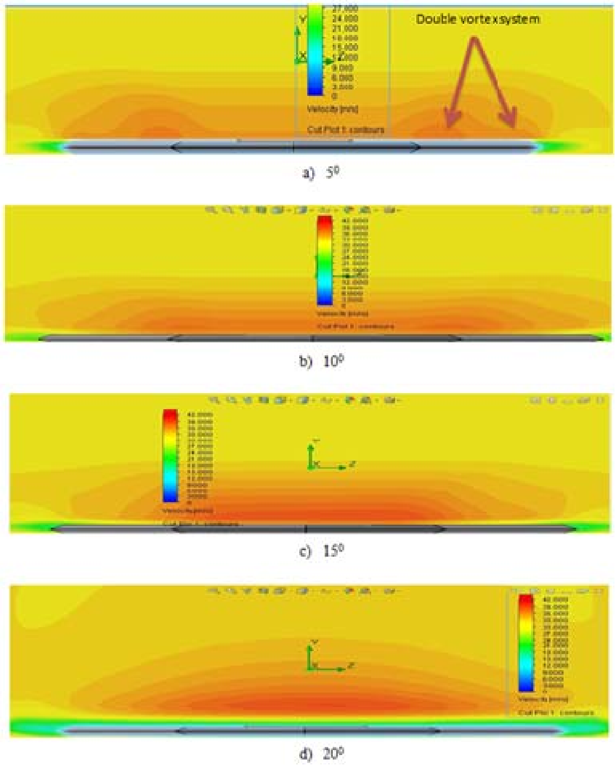

Figure 4.2: Front view contour of total velocity at 70% chord at Re = 1 x 10^6

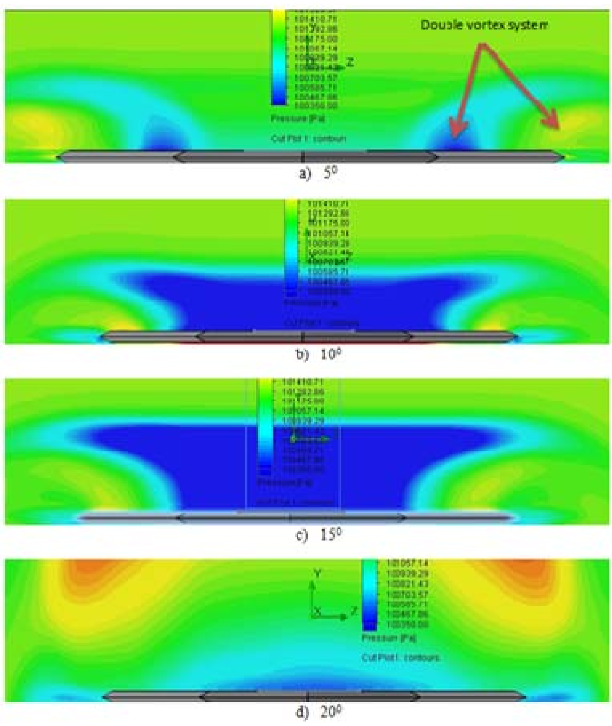

Figure 4.3: Front view contour of total pressure at 70% chord at Re = 1 x 10^6

Journal of Aeronautical
-Science and Engineering-, Vol.2

May 20, 2015

4.3 Aerodynamic Characteristics

As mention before, aerodynamic characteristic is the most important data for aircraft flight. Lift and drag coefficient will be one of determination of the performance of an aircraft. Equation 4.1 and equation 4.2 was used to gain those aerodynamic characteristics.

$$C_L = \frac{F_y}{\frac{1}{2}pV^2S} \tag{3}$$

$$C_D = \frac{F_x}{\frac{1}{2}pV^2S} \tag{4}$$

Table 4.2: Lift and Drag Coefficient of double delta wing using CFD

Reynolds Number	1×10^6		2×10^6	
Angle of attack (deg)	C_L	C_D	C_L	C_D
0	0.006668	0.065309	0.012645	0.044611
5	0.199926	0.107474	0.141193	0.098939
10	0.504066	0.139798	0.414291	0.151727
15	0.680203	0.212802	0.701801	0.196719
20	0.901917	0.41173	1.1046	0.380174

Figure 4.4 shows behaviors of lift coefficient react with angle of attack at different Reynolds number. Lift curve slop develop from 2×10^6 Reynolds number seems more higher compared to 1×10^6 Reynolds number. Figure 4.5 shows variation of drag coefficient at different angle of attack at one and two million of Reynolds number. This shows that Reynolds number did not give so much effect on drag coefficient.

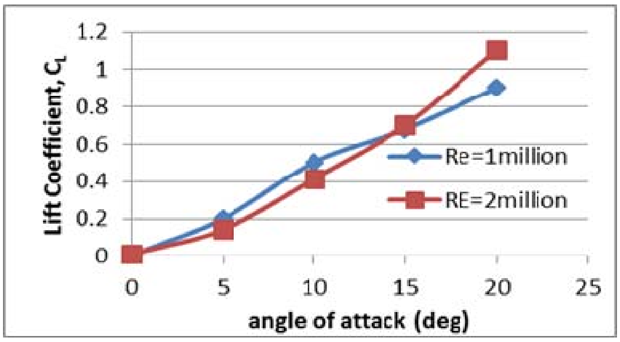

Figure 4.4: Lift coefficient at different Reynolds Number

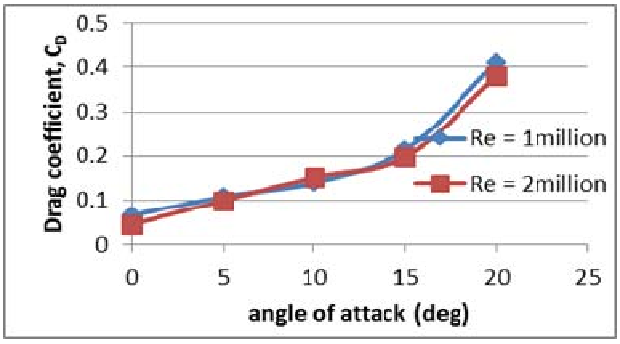

Figure 4.5: Drag coefficient at different Reynolds Number

4.4 Comparison Result with Wind Tunnel Testing

In wind tunnel experiments, six-component external balance is used to get the value of the aerodynamic loads such as lift, drag, side force, pitching moment, rolling moment and yawing moment. In this particular study, we only consider the lift and drag force since this two aerodynamic characteristic is our main concern.

Table 4.3: Comparison of lift coefficient (C_L) between CFD and Experimental

Angle of attack (deg)	CFD Result	EXP Result	ΔC_L
0	0.0066	-0.0641	1.103
5	0.1999	0.3485	0.426
10	0.5040	0.7188	0.298
15	0.6802	0.9073	0.250
20	0.9019	1.0781	0.613

Table 4.4: Comparison of drag coefficient between CFD and Experimental

Angle of attack (deg)	CFD Result	EXP Result	ΔC_D
0	0.0066	-0.0641	0.1876
5	0.1999	0.3485	0.2198
10	0.5040	0.7188	0.2688
15	0.6802	0.9073	0.0489
20	0.9019	1.0781	0.1401

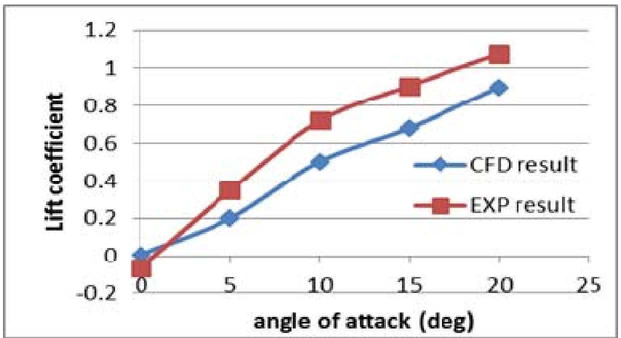

Figure 4.6: Comparison lift coefficient at Re = 1×10^6

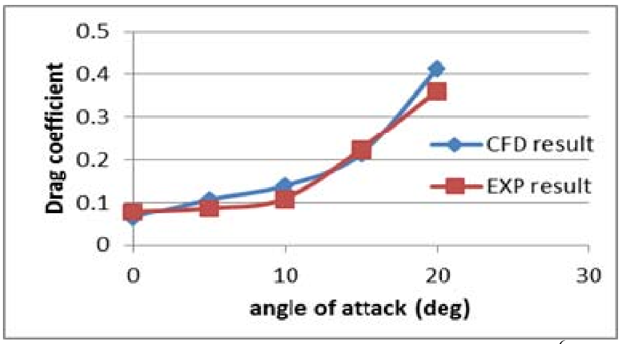

Figure 4.7: Comparison drag coefficient at Re = 1×10^6

Based on both figure for lift and drag coefficient for CFD result and wind tunnel result shows similar pattern and trends of graph. In addition, Table 4.2 and Table 4.3 show the value of error gain from CFD result compare to the wind tunnel testing. The highest percentage error in lift coefficient is at 0^0 angle of attack with almost 100 percent error. This is due to CFD result gain positive value while experimental gain negative value. However, well known that at 0^0 angle of attack, the lift coefficient is supposed to be zero and both of CFD and experimental result give value near to zero.

5.0 CONCLUSION

In this final year project, an aerodynamic study of double delta wing was performed by using Computational Fluid Dynamic (CFD) code Fluent. This study was focusing on aerodynamic characteristic of the model. Scope of this study is about literature review on CFD and double delta wing, simulation of CFD on double delta wing and comparison of the result from CFD with wind tunnel experiment.

A simulation by CFD has been carried out on a 65/25 degree of double delta wing at angles of attack ranging from 0 to 20 degree and at Reynolds number for one million and two million. A grid independent study was carried out to get an accurate result. It is shown that mesh model with over one million of element will be give better result. A mesh model can be varied from 200 thousand to 2 million of no of elements.

The error may happen due to less mesh quality in simulation process. Even a slight change in mesh quality will affect the simulation result. There are two types of mesh in CFD software that is structured mesh and unstructured mesh. The mesh model use in this simulation is unstructured mesh. This is because unstructured mesh consumes less time and much easier to handle compared to structured mesh. Another factor affecting the result between the CFD and experimental may come from human error while conducting the experiment.

It is shown that double delta wing can achieve high maneuverability and agility in its performance. Therefore, for further study, firstly, additional angle of attack for this model to see the characteristic since this study only simulate until 20^0 of angle of attack. From there, more result will be obtained for the agility of the wing itself. Secondly, highly recommended one can simply make a structured mesh of double delta wing model in order to gain higher quality of mesh model. Besides that, it is recommended to run the simulation with difference turbulence model such k-ε, spalart-allmaras, etc.

Then, for further studies of this double delta wing model, some configuration can be made such as instead of using sharp leading edge, a blunt leading edge can be used. Other than that, adding a control surface to the model such as flap to see how it will affect the aerodynamic characteristic of the model. Furthermore, it is also recommended to make a model of double delta wing with difference sweep angle combination such as $65^0/45^0$ or $70^0/40^0$ to compare and study the differences.

ACKNOWLEDGEMENTS

The authors would like to convey a great appreciation to Department of Aeronatical, Automotive and Ocean Engineering, Universiti Teknologi Malaysia, Malaysia for supporting this research.

REFERENCE

1. Shaw, C. T. (1992). Using Computational Fluid Dynamics. Prentice Hall.
2. John D. Anderson, J. (2011). Fundamental of Aerodynamics. New York, McGraw Hill, Inc.
3. S. Saha, M. (2012). Flow Visualization and CFD Simulation on 65^0 Delta Wing at Subsonic Condition. Jadavpur University, Kolkata, India.
4. Nettelbeck, C. (2008). Dynamic Analysis of a Double Delta Wing in Free Roll. School of Aerospace, Civil and Mechanical Engineering, (BE)
5. Breitsamter, A. F. C. (2012). "Turbulent and Unsteady Flow Characteristics of Delta Wing Vortex Systems."
6. Verhaagen N. G., Jenkins L. N., Kern S. B., and Washburn A. E. A study of the vortex flow over a 76/40-deg double-delta wing. AIAA-1992-279 33rd Aerospace Sciences Meeting and Exhibit, Reno NV, 1995.
7. Lu Zhiyong, Zhu Lirguo (February 2004), Study on Forms of Vortex Breakdown over Delta Wing
8. Luckring, J. M. (2010). A Survey of Factors Affecting Blunt Leading-Edge Separation for Swept and Semi-Slender Wings. AIAA 28th Applied Aerodynamics Conference. Chicago
9. John D. Anderson, J. (1995). Computational Fluid Dynamic, The Basics With Applications. United State of America, McGraw-Hill, Inc
10. Hesamodin Ebnodin Hamidi, M. R. (2011). Numerical Investigation of High Attach Angle Flow on $76^o/45^o$ Double Delta Wing in Incompressible Flow. World Academic of Science, Engineering and Technology.
11. N. Verma, D. S. S. (September 2012). "Spalart Allmaras Unsteady Flow Investigation Using Computational Fluid Dynamics." International Journal of Engineering Research & Technology (IJERT) **Volume 1**(Issue 7).