International Journal of Mechanical Engineering and Technology (IJMET)
Volume 09, Issue 13, December 2018, pp. 1555–1563, Article ID: IJMET_09_13_157
Available online at http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=09&IType=13
ISSN Print: 0976-6340 and ISSN Online: 0976-6359

NUMERICAL STUDY OF FLOWS AROUND THE EUROFIGHTER TYPHOON-LIKE FIGHTER AIRCRAFT ON SUBSONIC SPEEDS

Sutrisno
Department of Mechanical and Industrial Engineering, Faculty of Engineering, Universitas Gadjah Mada, Yogyakarta, Indonesia 55281

Petrisius Ginting
Department of Mechanical and Industrial Engineering, Faculty of Engineering, Universitas Gadjah Mada, Yogyakarta, Indonesia 55281

Setyawan Bekti Wibowo
Department of Mechanical Engineering, Vocational College, Universitas Gadjah Mada, Yogyakarta, Indonesia 55281

Sigit Iswahyudi
Department of Mechanical Engineering, Vocational College, Universitas Gadjah Mada, Yogyakarta, Indonesia 55281

Wega
Department of Mechanical and Industrial Engineering, Faculty of Engineering, Universitas Gadjah Mada, Yogyakarta, Indonesia 55281

ABSTRACT

Fighter aircraft had different strategic designs compared to passenger aircraft. Fighter aircraft should have good maneuverability so that its vortex dynamics analysis became important. Until now, research on fighter aircraft continued to develop, toward aircraft design sophistication. This study analyzed the fluid flow around a Eurofighter Typhoon-like aircraft to determine the aircraft performs on the changes in the angle of attack (AoA). This research was conducted using computational fluid dynamics (CFD) method because it had high accuracy and cost effectiveness. The role of research was very important especially in supporting combat aircraft operations for conditions that could not be simulated in wind tunnels or be too dangerous carried out in flight tests. From this study one could obtain the results of fluid flow analysis around the Eurofighter Typhoon-like aircraft in the form of Cl, Cd, Cl / Cd with respect to the AoA, graph of axial speed changes along the wing and canard vortex cores, graph of the wing and canard vortex cores at different angle of attack, surface pressure contour on the fuselage, and a graph of the pressure coefficient at 30% and 60% root wing chord and canard.

Keywords: Fighter aircraft, CFD, Eurofighter, *Vortex Core, Vortex Breakdown.*

Cite this Article: Sutrisno, Petrisius Ginting, Setyawan Bekti Wibowo, Sigit Iswahyudi and Wega, Numerical Study of Flows around the Eurofighter Typhoon-Like Fighter Aircraft on Subsonic Speeds, International Journal of Mechanical Engineering and Technology, 09(13), 2018, pp.1555–1563
http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=09&IType=13

1. INTRODUCTION

Along with the progress of the times, technology also continues to develop. To keep up with that development, research in the fighter industry continues to grow as well. The fighter planes in the design are different from passenger aircraft, far more complex because there are additional systems such as weapons control systems, avionics systems, as well as propulsion engine systems, radar systems, and aircraft structures that must be stronger but still agile in the air. To deal with this complexity, it is necessary to analyze the aerodynamics of fighter aircraft.

In fighter planes with a canard-wing configuration such as the Eurofighter Typhoon, the wing-canard configuration proved effective in stall delay caused by a delay in the main wing vortex breakdown due to canard [1]. The same thing was found in several other studies such as by R.M. Howard et al. [2], I.H. Tuncer [3], S. Hayashibara, et al.[4], S.B. Anderson [5], and J. Er-El et al. [6].

Current research and development of aircraft, many use wind tunnels, and Computational Fluid Dynamics (CFD) [7]. Research using CFD is quite accurate in a cost effective manner. This is done mainly in supporting combat aircraft operations with conditions that cannot be simulated in wind tunnels or too dangerous to be carried out in flight tests [8]. Because in this study the CFD method was chosen by using ANSYS FLUENT. In this study, the aerodynamic performance of the Eurofighter Typhoon fighter was investigated using the CFD method with the ANSYS FLUENT program based on existing research trends, namely plot parameters lift, drag and lift to drag ratio against angle of attack (AoA), plot axial velocity variations along the vortex core, contour plots surface pressure and plot the distribution of pressure coefficients along the wing span and canard.

In studies with the CFD method, the geometry model used is simplified, and because of its flow phenomena in aircraft with delta wings and wing-canard configurations, it can be considered symmetrical. Therefore it was decided to use a symmetrical geometry model to save grid usage. The symmetrical method was also carried out by other CFD researchers as done by O.J. Boelens et al. [8], S. Samimi et al. [9] and Chen, et al. [10] as a trend of air flow research using the CFD method.

Although the geometry model used has been simplified, there are still complex aircraft parts. For complex geometry, the solution will also depend on the shape of the grid, in this case, we will tend to develop a grid that is adjusted for the complexity of the geometry so that the shape and size of the grid will vary through the plane of flow. Making a grid for complex geometry is very crucial, its importance increases along with the dimensions of space. This step is the most significant in a three-dimensional CFD simulation [11].

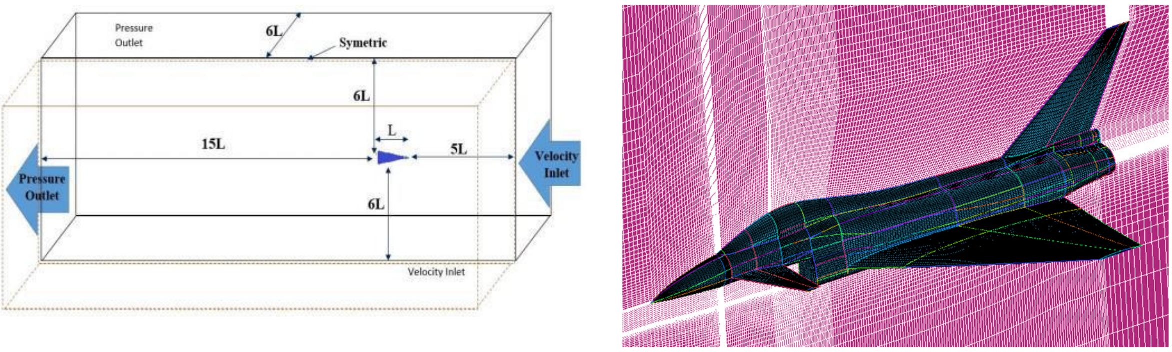
Making grids or meshing is the most important part of the CFD modeling process. Good grid quality, grid suitability with flow patterns, grid resolution right near the surface and so on are requirements of successful CFD simulations [12]. Computational domains are divided into several zones to produce structured grids. Structured grid produces a point with a unique index so that access to adjacent points can run efficiently which will speed up the computational process [12]. Structured grids also offer higher numerical accuracy and fewer data storage compared to unstructured grids [13].

In addition to the griding aspect, the turbulent flow physical model is very important, in engineering most flows are turbulent [14]. The modified Navier-Stokes equation, known as the Reynolds Average Navier Stokes or RANS equation describes the momentary variables / properties in the average value of fluctuating components for turbulent flow. Then these fluctuations are interpreted as statistical quantities at the average time value. Therefore all temporal scale turbulent fluctuations are included in the mathematical model of the average time in RANS [12].

Most delta wing studies with computational processes generally use the κ-ω turbulent model. As done by Soemarwoto et al. [15], in both studies using turbulent Shear Stress Transport (SST) κ-ω models because the turbulent Shear Stress Transport (SST) κ-ω model is considered to predict flow separation with higher accuracy and is the right choice for delta wing flow phenomena. All three have made comparisons of computational results and drawn the conclusion that the simulation results of CFD with turbulent models SST κ-ω showed adequate results, and there was conformity with the experimental results.

2. RESEARCH METHODS

The geometry model in this study was a Eurofighter Typhoon-like fighter with dimensions that were adjusted to the original size of 15.96 m and a wingspan of 10.95 m. To reduce computational burdens, geometry models were simplified and made in the form of half models in CAD software. Computational domains were also created in half the model with boundary condition and size as in Fig. 1a base on previous study [16]. This study uses a structured grid with a total grid of 8.3 million with its structured grid shown in Fig. 1b where mesh compaction is carried out around the aircraft body.



(a) Computational domain (b) Structured Grid

Figure 1 Computational domain, boundary condition and meshing structure Eurofighter Typhoon-like

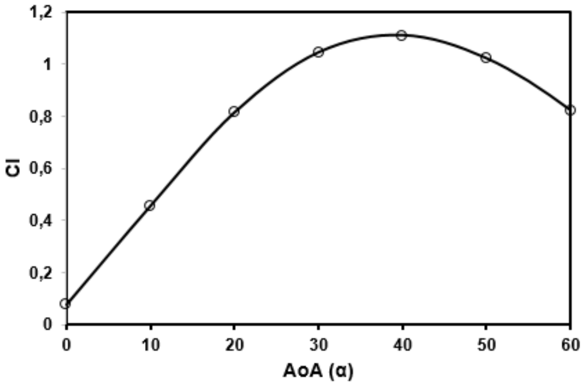
The computational process was carried out with FLUENT ANSYS with computational settings input summarized in Table 1. After the computation settings had been initialized, then the computational process was done with the number of iterations of 100 iterations. After the computing process had been completed, a post processing process was carried out. This process was the process of retrieving data using the ANSYS RESULT program. Some data was in the form of image and numerical data. Data in the form of numbers must first be processed using Microsoft Office Excel so that it can be presented in graphical form to facilitate the analysis.

Table 1 Summary of computational settings

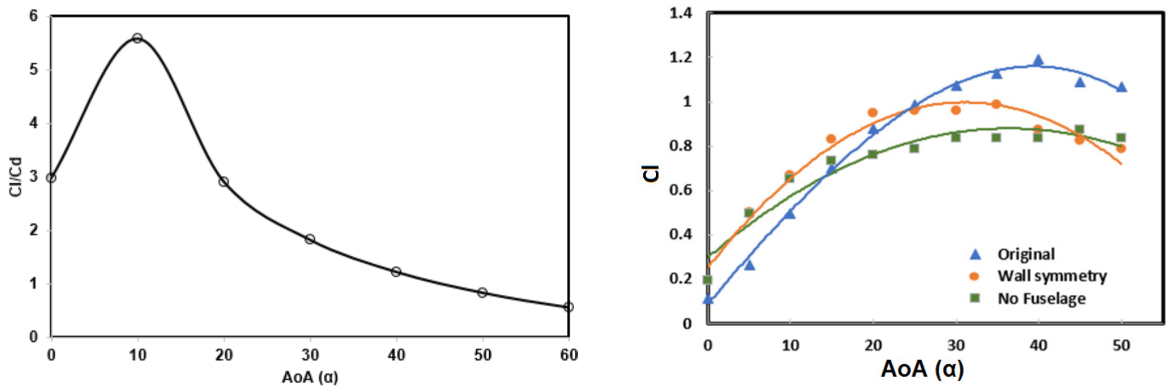
General	Solver Type	Pressure-Based
	Time	Steady
Models	Viscous Model	SST k-omega
Materials	Fluid	Air
Boundary Condition	Body	no-slip wall
	Velocity_Inlet	Velocity inlet (Magnitude & Direction)
		102.09 m/s magnitude
	Pressure_Outlet	Pressure Outlet
	Symmetry	Symmetry
Solution Initialization	Standard	From Velocity_inlet
Calculation	Number of Iteration	100

3. RESULT AND DISCUSSION

The collection of simulation data was in the form of lift coefficient (Cl), drag coefficient (Cd), lift / drag, plot of axial velocity variations along the vortex core, trajectory of wing and canard vortex cores, pressure contour on the plane surface, and distribution of pressure coefficient at 30% and 60% canard and wing root chords.



(a) Cl (b) Cd

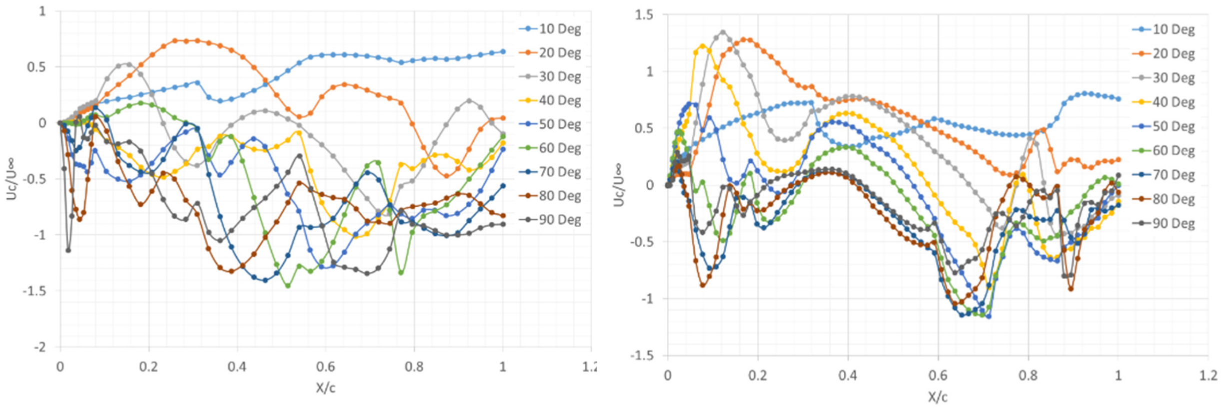


(c) Cl/Cd (d) Chengdu J-10-like fighters

Figure 2 The graph of Cl, Cd, Cl/Cd with respect to AoA comparison between Eurofighter Typhoon-like and Chengdu J-10-like fighters at water tunnel.

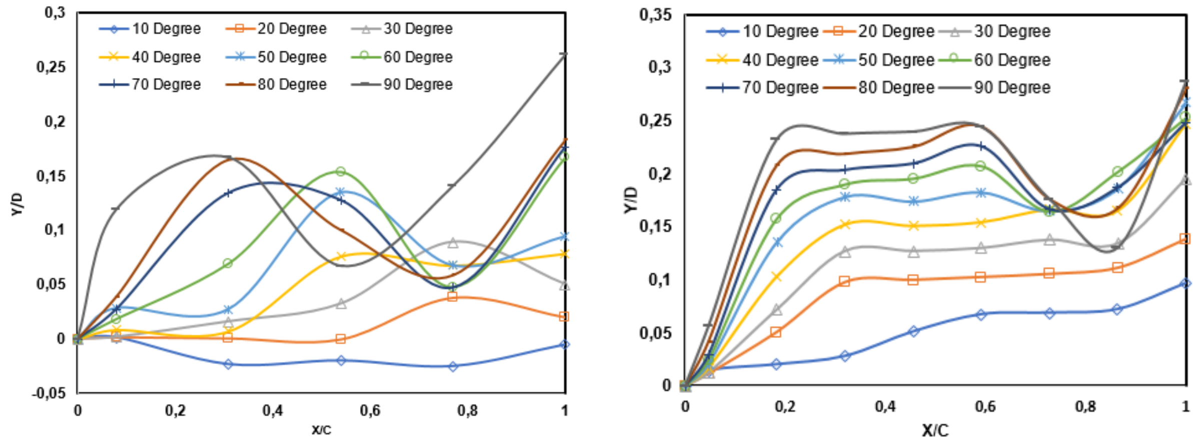
Cl and Cl/Cd plot for the AoA for the Eurofighter Typhoon-like fighter model is shown in Fig. 2, compared to Chengdu J-10-like fighters at CFD simulations and water tunnel experiment. Axial Speed Variations throughout the canard vortex core was displayed in Fig. 3. Canard and Wing Vortex Core Trajectory of a Eurofighter Typhoon-like fighters were taken in the form of the trajectory of high vortex core and spanwise vortexcore. The heights of the canard and wing vortex core at various AoAs are shown in Fig. 4.

The graph of Cp distribution along the span of 30% and 60% wing chord of a Eurofighter Typhoon-like are presented at Fig. 5. Body pressure contour above the wing and canard of a Eurofighter Typhoon-like fighters is presented in Fig. 6.



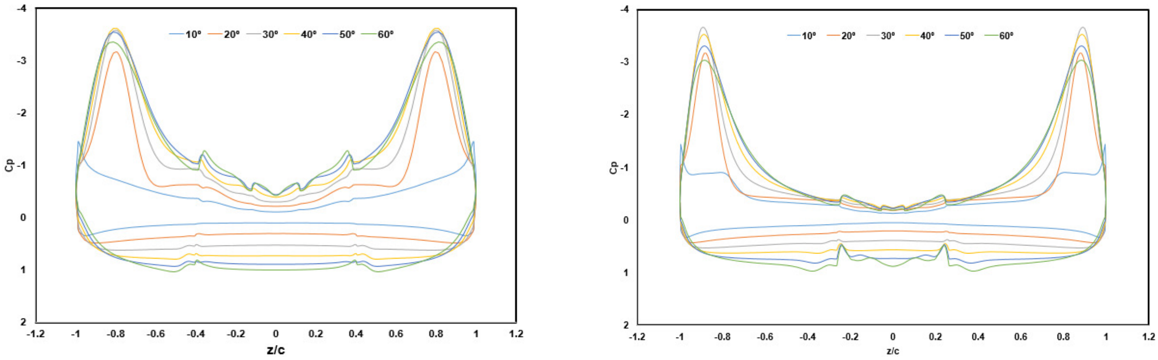
(a) Canard Vortex Core (b) Wing Vortex Core

Figure 3 Graph of U_c/U_∞ variation along canard vortex core of a Eurofighter Typhoon-like fighters, with respect to x/c for different of an angle of attack.



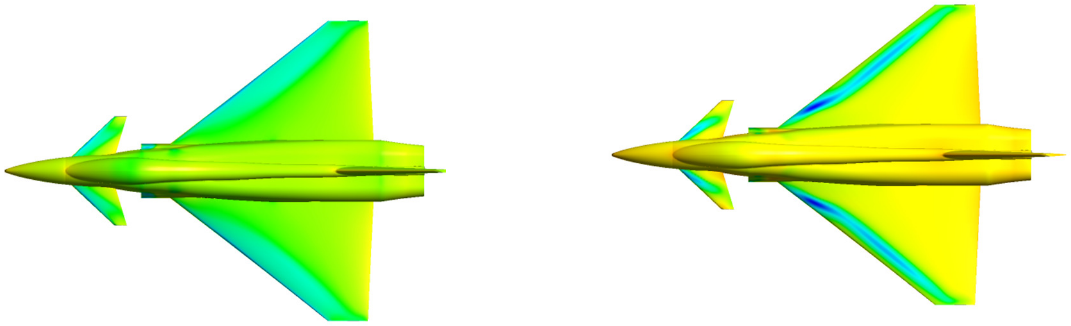
(a) canard vortex core (b) wing vortex core

Figure 4 Graph of the height of canard and wing vortex core of a Eurofighter Typhoon-like fighters at various angles of attack

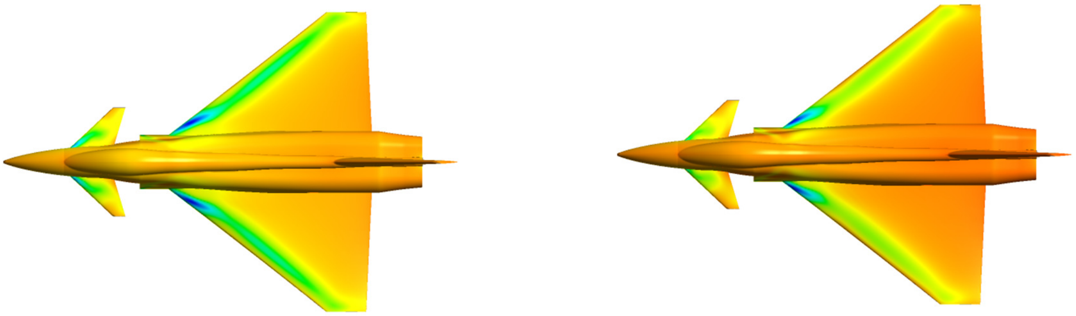


(a) 30% *wing chord* (b) 60% *wing chord*

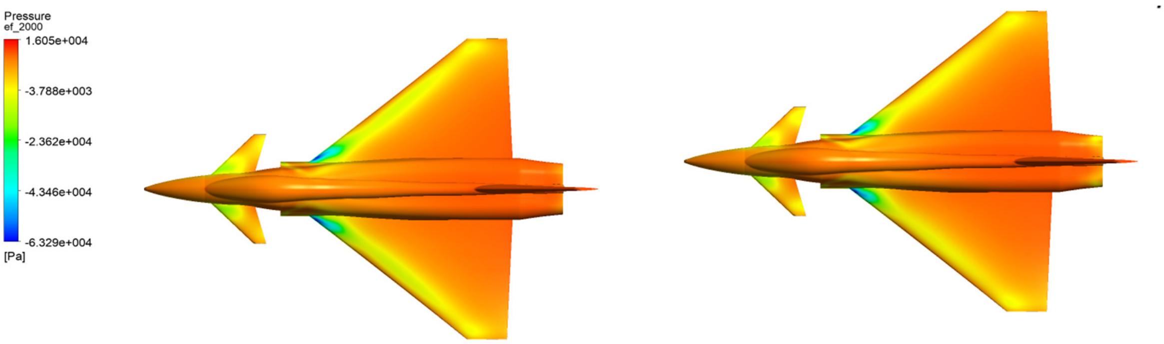
Figure 5 Graph of Cp distribution along the span of 30% and 60% wing chord of a Eurofighter Typhoon-like fighters



(a) AoA (α) 10^0 (b) AoA (α) 20^0



(c) AoA (α) 30^0 (d) AoA (α) 40^0



(e) AoA (α) 50^0 (f) AoA (α) 60^0

Figure 6 Body pressure contour at a various angle of attack

Based on the results of this research, it shows that the aerodynamic performance of Eurofighter Typhoon-like aircraft is different for each change in angle of attack. Where the values of Cl, Cd, and Cl / Cd have a form of change each where the value of Cl continues to increase as the AoA increases to reach the stall at an angle of 40^0 with a value of 1.11 and then the value decreases. Cl value has a slope of 2.1659 / rad, so it can be concluded that Eurofighter Typhoon-like planes have better maneuverability compared to aircraft that have a sloping Cl curve for the same maximum Cl value. From the value of Cl, it can be seen that the Eurofighter Typhoon-like aircraft has the ability to maneuver in the air up to the 40^0 AoA. After reaching the stall point, the maneuvering ability decreases if the AoA continues to rise. The Cd value continues to increase as the angle of attack increases to an angle of 90^0. From the Cd value this reduces aircraft aerodynamic performance so that overall aircraft performance can be seen from the graph of the value of Cl/Cd where the value of Cl/Cd also increases to the angle of attack 10^0 which is the point of maximum aerodynamic performance that is equal to 5.57, and then the value decreases as the AoA increases.

To visualize fluid flow in Eurofighter Typhoon-like aircraft models also have differences between the wing and canard parts. This difference is due to different canard configurations and wings, where the canard configuration is tilted down while for wing configurations it is the same as the wing configuration in general. As a result of the different configurations, several parameters of flow analysis also have different characteristics between wings and canard where the effect of the vortex core phenomenon in the canard region is greater on the fuselage of the plane than canard itself because the location of vortex cores tends to be closer to the fuselage than a canard. As for wings, the effect of the vortex core phenomenon is greater on the wing than the fuselage although the effect on fuselage increases with increasing AoA due to the movement of the vortex core that is getting closer to the fuselage. However, the results of this increase are not very significant. This can be seen by comparing the data presented in Fig. 6 where the data shows differences in the simulation results on wings and canards.

From Fig. 5 it can also be seen that the pressure drop due to the vortex core phenomenon decreases due to the increase in the AoA indicated by the moving suction location towards the leading-edge along with the increase in the AoA. This is caused by the height of the vortex core increasing along with the increase in the AoA shown in Fig. 4. As a result of this movement, the effect of the vortex flow velocity on the surface also decreases so that the reduction in pressure is also reduced.

4. CONCLUSION

In the Eurofighter Typhoon-like fighter model, a phenomenon of vortex core and vortex breakdown in both the canard and wing was identified. Eurofighter Typhoon-like aircraft model stalled at 40^0 AoAs with a maximum Cl value of 1.11 with a slope of the Cl curve of 2.1659 / rad. Cd value increased with increasing angle of attack. The biggest Cd value was at the 90^0 AoA which was equal to 1.6836. The ratio of lift and drag increased with increasing AoA until it reached a maximum value of 10^0 which was equal to 5.5683 or the lift value was 5.57 times the drag value. After reaching its maximum value, the lift and drag ratio continued to decrease as the AoA rises to 90^0.

The maximum axial speed increased in canard vortex core occurs at the 20^0 AoA, which was equal to 0.7331 times the free stream speed. While on the wing occured at 30^0 AoA, which was equal to 1.346 times the free stream speed. In canard, the effect of the AoA on the vortex core trajectory could not be concluded as a whole but can only be discussed at each point at the x/c location. As for the vortex core trajectory on the wing, it could be concluded that the higher the AoA, the vortex core trajectory was also higher and widens away from the fuselage. The difference in the results of the visualization of the track was caused by a different configuration between the canard and the wing where the canard extended downward while the wings extended to the side.

The wing and canard suction area were indicated by the surface pressure contour of the Eurofighter Typhoon-like aircraft. Where from the surface pressure contour it could be seen that the suction area approached the leading-edge as the angle of attack increased. Graph of the pressure coefficient (Cp) at the location of 30% chord and 60% wing and canard chord showed the presence of suction area and suction peak value. Suction peak 30% canard chord occured at 70^0 AoA with a value of -5.6147, 30% suction peak wing chord occueds at 40^0 AoA with a value of -3.6254, suction peak 60% canard chord occured at 40^0 AoA with the value was -3.9263, 60% suction peak wing chord occured at 40^0 AoA with a value of -3.6671.

ACKNOWLEDGMENTS

The authors would like to express heartfelt gratitude to Prof. Indarto for a fruitful session, useful suggestions, and collaboration. We appreciate the help of our students Zainuri Anwar, Wega, Handika, Caesar, and Yogi, and the lab staff members, Ponimin and Wajiono, for giving their help in construction work and conducting data management, which we gratefully acknowledged.

NOMENCLATURE

α = angle of attacks (AoA) degree
c = chord
D = aircraft height
p = pressure
x = chordwise coordinate
y = vertical coordinate
z = spanwise coordinate
y^+ = dimensionless wall distance
Cp = pressure coefficient
Cl, Cd = Coefficient of lift, drag
U_c, U_∞ = local, free stream velocity (m/s)

REFERENCES

[1] B. B. Gloss and L. W. McKmney, “Canard-Wing Lift Interference Related to Maneuvering Aircraft at Subsonic Speeds,” 1973.

[2] R. M. Howard and J. Kersh, Jr., “Effect of canard deflection on enhanced lift for a close-coupled-canard configuration,” in 9th Applied Aerodynamics Conference, 1991.

[3] I. H. Tuncer and M. F. Platzer, “Computational Study of Subsonic Flow over a Delta Canard-Wing-Body Configuration,” J. Aircr., vol. 35, no. 4, pp. 554–560, Jul. 1998.

[4] S. Hayashibara, R. Y. Myose, and L. S. Miller, “The effect of a 70 deg swept canard on the leading-edge vortices of a 70 deg swept delta wing during dynamic pitching,” no. January, pp. 1–9, 1997.

[5] A. JD, “Anderson J.D.- Computational fluid dynamics. The basics with applications.pdf.” 1995.

[6] J. Er-El and A. Seginer, “Vortex Trajectories and Breakdown on Wing-Canard Configurations,” J. Aircr., vol. 22, no. 8, pp. 641–648, Aug. 1985.

[7] Subagyo, “Numerical simulation of unsteady viscous flow around airship using vortex method,” J. Teknol. Dirgant. Vol. 8 No. 2 Desember 2010125-135 unsteady, vol. 8, pp. 125–135, 2010.

[8] O. J. Boelens, “CFD analysis of the flow around the X-31 aircraft at high angle of attack,” Aerosp. Sci. Technol., vol. 20, no. 1, pp. 38–51, 2012.

[9] S. Samimi, A. R. Davari, and A. M. R. Soltani, “Canard-Wing Interactions in Subsonic Flow,” Trans. Mech. Eng., vol. 37, no. M2, pp. 133–147, 2013.

[10] M. Chen, P. Liu, H. Guo, and Q. Qu, “Effect of sideslip on high-angle-of-attack vortex flow over close-coupled canard configuration,” J. Aircr., vol. 53, no. 1, pp. 217–230, 2016.

[11] C. Hirsch, Ed., Numerical Computation of Internal &Amp; External Flows: Fundamentals of Numerical Discretization. New York, NY, USA: John Wiley & Sons, Inc., 1988.

[12] S. Saha and B. Majumdar, “Flow Visualization and CFD Simulation on 65° Delta wing at Subsonic Condition,” Procedia Eng., vol. 38, pp. 3086–3096, 2012.

[13] A. M. Alakashi and D. I. B. Ba, “Comparison between Structured and Unstructured Grid Generation on Two Dimensional Flows Based on Finite Volume Method (FVM),” Int. J. Mining, Metall. Mech. Eng. Vol. 2, Issue 2 ISSN 2320–4060, vol. 2, no. 2, pp. 97–103, 2014.

[14] S. B. Wibowo, Sutrisno, and T. A. Rohmat, “An evaluation of turbulence model for vortex breakdown detection over delta wing,” Arch. Mech. Eng., vol. LXV, no. 3, pp. 399–415, 2018.

[15] B. I. Soemarwoto and O. J. Boelens, “Simulation of vortical flow over a slender delta wing experiencing vortex breakdown Simulation of vortical flow over a slender delta wing experiencing vortex breakdown,” 2003.

[16] S. Wibowo, S. Sutrisno, and T. Rohmat, “Study of Mesh Independence on the Computational Model of the Roll-up Vortex Phenomena on Fighter and Delta Wing Model,” Int. J. Fluid Mech. Res., vol. 46, no. 2, p. in press, 2018.