

CFD ANALYSIS OF AN RC AIRCRAFT WING

¹SHREYAS KRISHNAMURTHY, ²SURAJ JAYASHANKAR, ³SHARATH V RAO, ⁴ROCHEN KRISHNA T S, ⁵SHANKARGOUD NYAMANNAVAR

^{1,2,3,4,5}Department of Mechanical Engineering, JSS Academy of Technical Education, Bangalore.
E-mail: shreyaskrishnamurthy@gmail.com, surajjayashankar@gmail.com, sharath0692@gmail.com,
rochenkrishnats@gmail.com, nyamannavar@gmail.com

Abstract- Computational fluid dynamics (CFD) analysis of an aircraft plays a vital role in providing an ideal design of the aircraft. The qualitative and quantitative characterization of the wing provides useful information to verify the wing selection and design prior to the time consuming fabrication of aircraft. A designed wing geometry was analyzed and modified; further analysis was carried out on the modified wing for a comparative study. Thus giving a clear comparative view of how minute changes affect the flow characteristics of the aircraft. For this very purpose, a virtual wind tunnel model was created and CFD analysis was carried out at various angle of attacks for each wing separately. The lift, drag, stall angle and lift-to-drag ratio of each wing were determined and the comparative study of these showed how minute changes to the wing improves its overall flow characteristics.

Keywords- Computational Fluid Dynamics, Virtual wind tunnel model, Modified wing, Comparative study.

I. INTRODUCTION

The shape and design of an aircraft can dramatically influence how the aircraft is handled and controlled. The stability and control derivatives are essential for flight simulation and handling qualities. Wind tunnel test are a common method that results in derivatives that are highly accurate. The problem with wind tunnel tests is that they are very expensive and can be extremely time consuming. Also, the wind tunnel models are scaled down to fit in the tunnel, and this can have a dramatic change on the results, since the results do not always scale up as easily. Air will flow over a smaller body differently than a larger body, due to the changes in flow characteristics.

Over the past couple of decades, computer simulation has become much more prevalent. Current CFD software's are is much more accurate than they once were and are is becoming more user-friendly, but still the mesh generation for a model can be difficult and requires a great deal of experience. These software's can be used over and over again. Also, many different test cases can be run to determine flying qualities in various situations. There are also several different programs that are readily available that can produce highly reliable results. With the use of CFD, the cost and time required for prototyping has reduced drastically over the years, thereby accelerating the design and development of modern aircrafts.

CFD is now a proven method or tool for simulating and studying fluid flows in the subjects involving fluid, heat and mass transfer in different conditions. CFD basically works by solving the governing equations of fluid dynamics i.e. the Navier-Stokes equations. Wherein a system of coupled nonlinear

partial differential equations are numerically solved as they are very difficult to solve analytically. CFD helps engineers to find a nearly accurate approximate solution to the governing equation for a range of fluid flow problems.

In the present study, CFD approach for analysis has been adopted to understand the aerodynamic characteristics of a Vertical Take-off and Landing (VTOL)/Hybrid RC aircraft. A VTOL aircraft is one that can hover, take off, and land vertically. This classification includes fixed-wing aircraft as well as helicopters and other aircraft with powered rotors and tilt rotors.

II. CFD APPROACH & METHODOLOGY, STRUCTURE AND EXPERIMENTAL SETUP

A. CFD Approach & Methodology

Development of aircraft geometry involves derivation of geometrical equations of the aircraft. Some parameters such as wing area, sweep angle, taper ratio for each section, span and chord for various span wise locations must be determined. These design parameters are then translated into a 3D model in CATIA solid modeling software, to get the virtual model of an RC aircraft wing.

Second stage involves conversion from three dimensional CATIA CAD model into CFD element using ICEM CFD to create discretized model for CFD analysis. Then, the discretized model was exported to CFX for the analysis. Airflow over the wing was analyzed by defining appropriate boundary conditions. Pressure distribution on the surface of the wing of aircraft was estimated. The surface pressure

distribution data was used for calculations of aerodynamics characteristics of wing such as coefficient of lift ' C_L ' and coefficient of drag ' C_D ' at various angle of attack ' α '. Flow visualization module in the Post Processor of CFX was used to locate the critical area with possible turbulence reduction.

B. Structure

The structure of the wing of the aircraft plays a major role in determining the flow characteristics. The specifications of the wing are as follows:

- i. Airfoil – GOE523
- ii. Semi wing span ' S ' – 0.74m
- iii. Surface area ' A ' – 0.2195m²
- iv. Root chord ' C_r ' – 0.16m
- v. Tip chord ' C_t ' – 0.07m
- vi. Cruise speed/velocity ' v ' – 20m/s

The wing is rectangular from the root to 0.4 m of semi wing span, the wing sweep starts from 0.4m and ends with a tip chord length of 0.07m. The wing has a dihedral angle of 3° from 0.4 m semi span (i.e. from the point where the sweep begins).

Since the wing used was that of a VTOL/Hybrid aircraft, the wing had to accommodate a beam, due to which a rectangular cut out had to be made. The initial CFD analysis of the wing (Prototype-I) showed high turbulence near the cutout thus few modifications were made. The modified wing profile (Prototype-II) was subjected to further simulations. The RC aircraft under study is shown in Fig. 1.

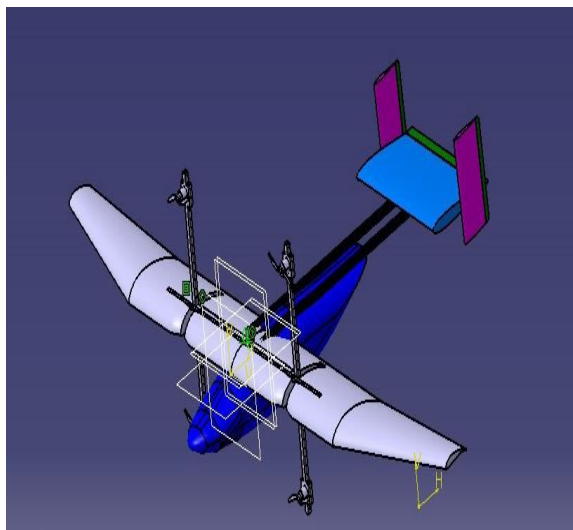


Fig. 1 - CATIA model of the VTOL/Hybrid Aircraft.

For Prototype-I, the position of the cutout for the mechanism to be placed is at a distance of 0.2m from root with a rectangular cross-section of 16mm. The wing has a semicircular cutout at 0.2 m from root with a radius of 0.10m whose center point is at a distance of

0.08m from aerodynamic center. Fig. 2 shows the wing of Prototype-I.

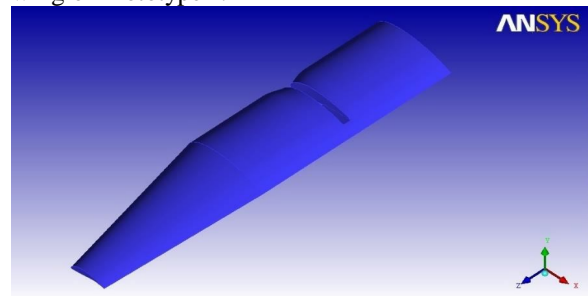


Fig. 2 - Wing structure of Prototype-I.

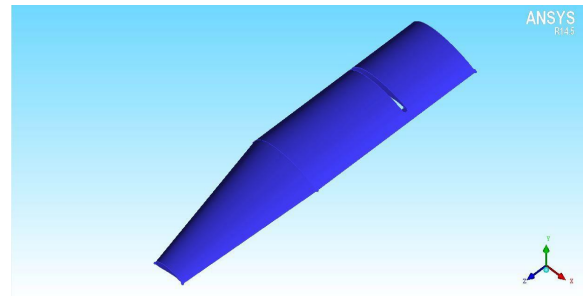


Fig. 3 - Wing structure of Prototype-II.

For Prototype-II, the position of the cutout for the mechanism to be placed was changed to a distance of 0.16m from root with a rectangular cross-section of 10mm. Fig. 3 shows the wing of Prototype-II.

C. Experimental Setup

A low speed wind tunnel was created to test the wing in subsonic conditions. This wind tunnel designed, i.e. domain was of a cuboid shape with dimensions 3.2 m x 3.7 m x 3.2 m. The tests were conducted at mean sea level conditions, i.e. boundary conditions, taking an air speed of 20 m/s at various angle of attacks between 0° to 20°. Only one half of the wing was considered due to its symmetric nature so as to reduce computational effort. A 3D unstructured tetrahedral mesh was utilized for computing the flow around the model as shown in Fig. 4. Unstructured mesh was appropriate due to the complexity of the model. The advantages of the unstructured mesh are shorter time consumption in grid generation for complicated geometries and the potential to adapt the grid to improve the accuracy of the computation. After the meshing process, the mesh was examined for quality.

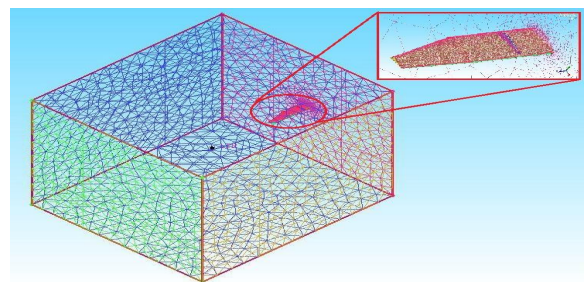


Fig. 4 - Cuboid shaped wind tunnel and the wing in discretized form.

III. RESULT AND DISCUSSION

The simulation results for initial model (Prototype-I) and e and the modified wing (Prototype-II) were compared. The discussions were focused on the aerodynamics characteristics which include drag coefficient 'C_D', lift coefficient 'C_L', and lift-to-drag ratio 'L/D'. In addition, the pressure coefficient contours, temperature contours and velocity contours were also analyzed at various angle of attacks 'α'.

Table 1. shows the values of Lift 'L', Drag 'D', Coefficient of lift 'C_L', Coefficient of drag 'C_D' and lift-to-drag ratio 'L/D' or 'C_L/C_D' for various angle of attacks of Prototype I and Prototype-II. We can clearly see an increased lift forces for Prototype-II with increasing angle of attack, similarly we also observe decrease in the drag forces with increasing angle of attack.

The maximum increase in lift is 3N at α = 14°, implying that the aircraft can carry an extra load of around 300g when compared to Prototype-I. The values of C_L and C_D were obtained using (1) and (2).

$$C_L = \frac{2L}{\rho \cdot v^2 \cdot S} \quad (1)$$

$$C_D = \frac{2D}{\rho \cdot v^2 \cdot S} \quad (2)$$

Table I. Result Table.

	α	L (N)	D (N)	CL	CD	CL/CD
Prototype I	0	20.9 2	2.20	0.4 2	0.0 4	9.50
	2	25.1 0	2.56	0.5 1	0.0 5	9.77
	4	28.9 5	3.04	0.5 9	0.0 6	9.49
	6	32.5 4	3.62	0.6 6	0.0 7	8.98
	10	38.6 5	4.95	0.7 8	0.1 0	7.80
	14	43.1 2	6.45	0.8 8	0.1 3	6.68
	16	44.7 1	7.22	0.9 1	0.1 4	6.18
	18	45.3 6	8.01	0.9 2	0.1 6	5.65
	20	45.1 9	8.83	0.9 2	0.1 8	5.11
Prototype II	0	21.8 0	2.23	0.4 4	0.0 4	9.73
	2	26.5 1	2.57	0.5 4	0.0 5	10.28
	5	32.9 0	3.29	0.6 7	0.0 6	9.99

10	41.2 6	4.78	0.8 4	0.0 9	8.61
14	46.1 1	6.24	0.9 4	0.1 2	7.37
16	47.4 7	6.96	0.9 6	0.1 4	6.81
18	47.6 8	7.69	0.9 7	0.1 5	6.19
20	47.0 4	8.39	0.9 6	0.1 7	5.60

D. Lift Coefficient C_L Analysis

Fig. 5 shows the variation in the lift curves of Prototype-I and Prototype-II, i.e. lift coefficient values with varying angle of attacks. Here it is observed that the Prototype-II gives a better performance and has higher load carrying capabilities for the same power input, i.e. increased lift force. From Fig. 5, we also find the stall angle of the aircraft to be 18°.

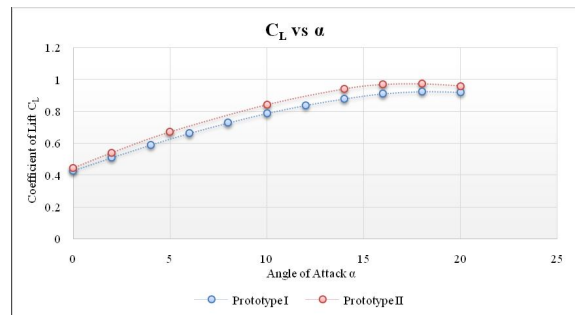


Fig. 5 – Lift coefficient C_L versus angle of attack α.

E. Drag Coefficient, C_D Analysis

Fig. 6 shows drag coefficient, C_D versus angle of attack, α. From the graphs, Prototype I & II do not show any variation in the value of drag coefficients up to 6° angle of attack, after which the drag coefficient values of Prototype-II shows gradual decrease with increasing angle of attack.

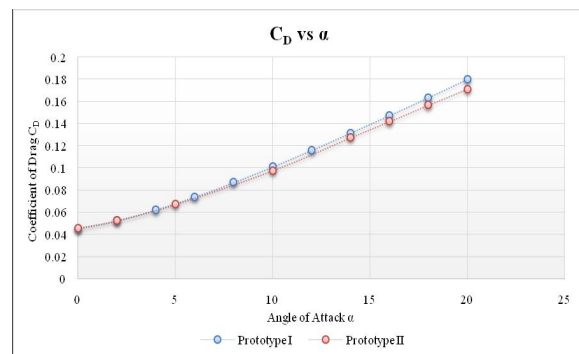


Fig. 6 - Drag Coefficient C_D at various angle attacks α

F. Lift-To-Drag Ratio C_L/C_D Analysis

The estimated values of C_L/C_D were plotted against α and graph is shown in Fig. 7. Prototype-II exhibited higher values of L/D ratio for various angle of attacks. The maximum L/D Ratio is at 2° angle of attack for both the prototypes, but the ideal range of angle of

attack for flight is in between 0° - 4° for Prototype-I and it ranges from 0° - 5° for prototype-II.

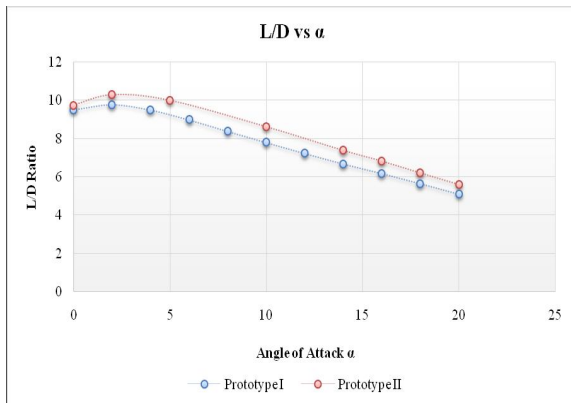


Fig. 7 - Lift-to-drag ratio, C_L/C_D at various angle attack, α .

G. Velocity Contours

The velocity contours obtained helped in determining the turbulence in the fluid flow. The velocity contours at 0° & 18° angle of attacks for Prototype I and II are shown figure Fig. 8(a), 8(b) 9(a) and, 9(b).

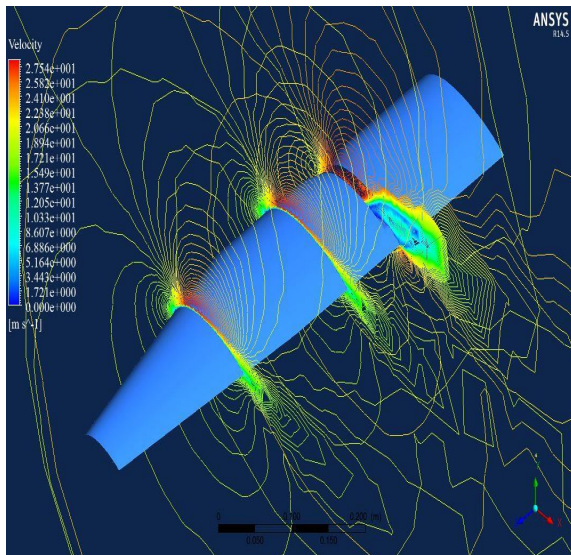


Fig. 8(a) - Velocity contours over Prototype-I at $\alpha=0^{\circ}$.

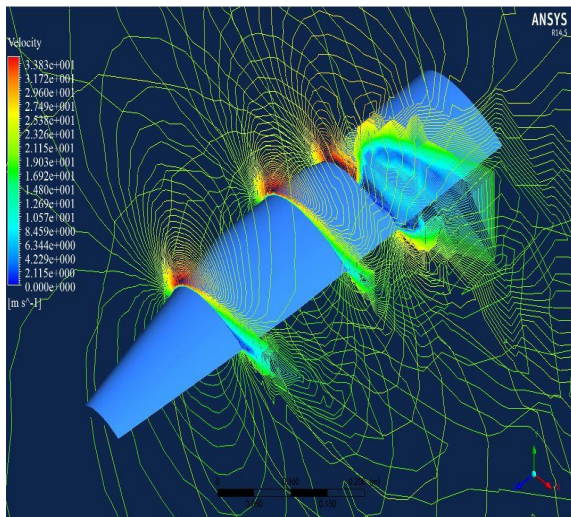


Fig. 8(b) - Velocity contours over Prototype-I at $\alpha=18^{\circ}$.

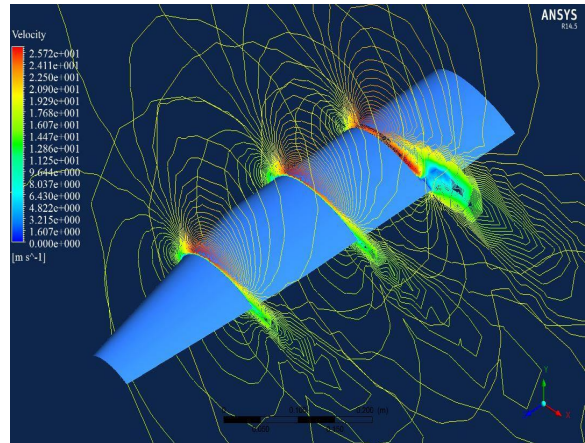


Fig. 9(a) - Velocity contours over Prototype-II at $\alpha=0^{\circ}$.

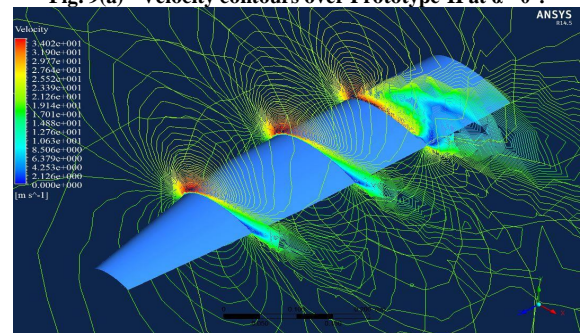


Fig. 9(b) - Velocity contours over Prototype-II at $\alpha=18^{\circ}$.

These velocity contours show an increase in the turbulence of the flow due to the flow separation created with increasing angle of attack. This flow separation can be observed by comparing Fig. 8(a) and 8(b) for Prototype-I and Fig. 9(a) and 9(b) for Prototype-II. As turbulence increases with increasing angle of attack, the wing stalls at a particular point, in this case it stalls at $\alpha=18^{\circ}$.

H. Pressure Coefficient Contours

Fig. 10 (a & b) below show pressure coefficient contour of top surface for Prototype-I at 20 m/s at 0° and 18° angle of attack. When the angle of attack, α increases the lower pressure coefficient is developed on the upper surface. The high intensity blue area located on the upper surface suggests that high lift is generated. Prototype-II also showed similar changes in pressure contours from 0° to 18° angle of attack, evident from Fig. 11(a & b).

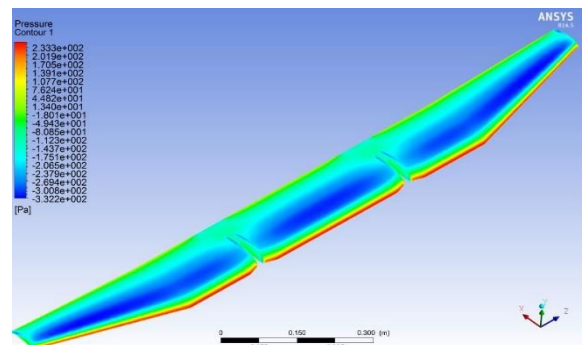


Fig. 10(a) - Pressure contours at $\alpha=0^{\circ}$ for Prototype-I.

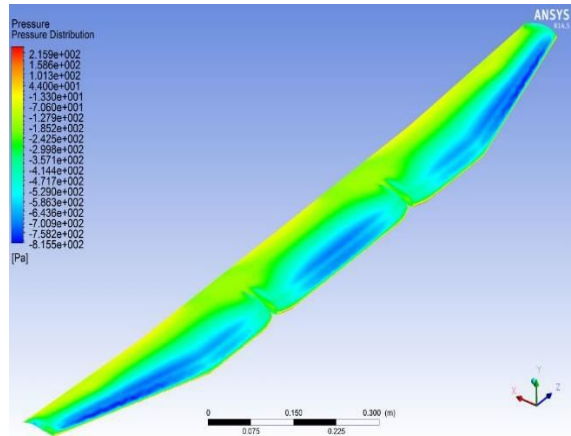


Fig. 10(b) – Pressure contours at $\alpha=18^\circ$ for Prototype-I.

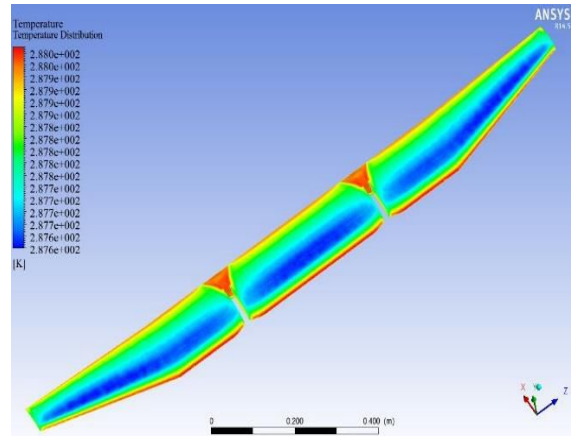


Fig. 12(a) - Top surface temperature contours at $\alpha=0^\circ$ for Prototype-I.

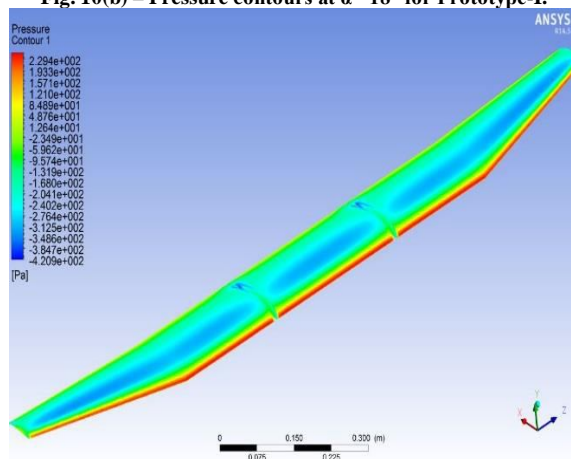


Fig. 11(a) – Pressure contours at $\alpha=0^\circ$ for Prototype-II.

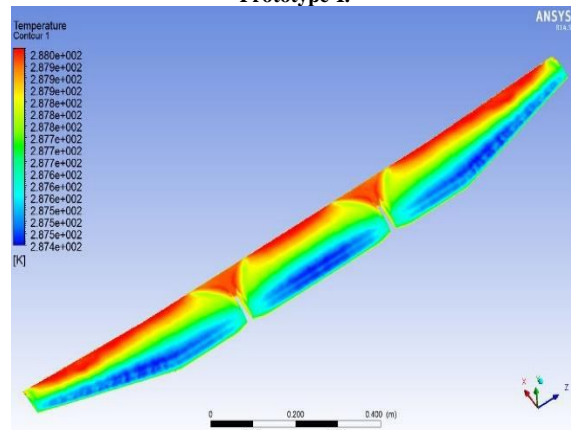


Fig. 12(b) - Top surface temperature contours at $\alpha=18^\circ$ for Prototype-I.

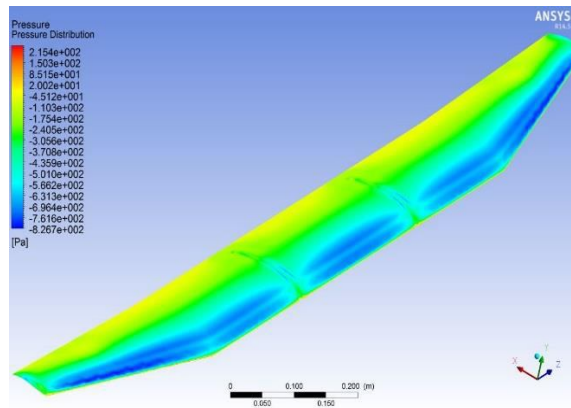


Fig. 11(a) – Pressure contours at $\alpha=18^\circ$ for Prototype-II.

Prototype-II shows higher pressure regions when compared to that of Prototype-I at 0° as well as 18° angle of attack. At high angle of attack α , lift is still generated, but most of the total force is directed backward as drag.

I. Temperature Contours

Fig. 12(a) and 12(b) shows the top surface temperature contours for Prototype-I at 20 m/s at 0° and 18° angle of attack. Since pressure is directly proportional to temperature, similar variation in temperature is also observed for both the Prototypes.

CONCLUSION

The analysis played a significant role in improving the wing design of the VTOL/Hybrid Aircraft. Modification was done on the wing after the preliminary analysis, thus shortening the time-cycle before actually coming out with the optimum design. With the comparative analysis of the designed wing of Prototype I and II, the wing design of Prototype-II proved to provide a superior performance than that of the later. The implemented changes in the wing after the analysis of Prototype-I played a major role in increasing the lift to a maximum of 3 N, and at the same time decreasing the drag forces. The analysis of the wing helped in significantly predicting a better performance and reduced power consumption of the aircraft due to reduced drag and increased lift forces.

REFERENCES

- [1] Wirachman Wisnoe, Rizal Effendy Mohd Nasir, Wahyu Kuntjoro, and Aman Mohd Ihsan Mamat, "Wind Tunnel Experiments and CFD Analysis of Blended Wing Body (BWB) Unmanned Aerial Vehicle (UAV) at Mach 0.1 and Mach 0.3", 13th International Conference on Aerospace Sciences & Aviation Technology, ASAT- 13, May 26 – 28, 2009.

- [2] William Runge, "Characterizing the Aerodynamics of a UAV Wing", Intermediate Fluid Mechanics, Prof. Luo, 12/18/2008.
- [3] Benjamin Sweeten, "CFD analysis of UAVs using VORSTAB, FLUENT, and Advanced Aircraft Analysis software", Aerospace Engineering, Graduate Faculty of the University of Kansas, April 27, 2010.
- [4] Tholudin Mat Lazim, Shabudin Mat, Huong Yu Saint, "Computational Fluid Dynamic Simulation (CFD) and Experimental Study on Wing-external Store Aerodynamic Interference of a Subsonic Fighter Aircraft", Acta Polytechnica Vol. 44 No. 2/2004.
- [5] L. A. Schiavetta, O. J. Boelens, W. Fritz, "Analysis of Transonic Flow on a Slender Delta Wing Using CFD", American Institute of Aeronautics and Astronautics.
- [6] M. Dinesh, Premkumar P S, J.S.Rao, C. Senthilkumar, "Performance Analysis of Winglets Using CFD Simulation", Altair Technology Conference, 2013.
- [7] M. A Azlin, C.F Mat Taib, S. Kasolang and F.H Muhammad, "CFD Analysis of Winglets at Low Subsonic Flow", Proceedings of the World Congress on Engineering 2011 Vol I, July 6 - 8, 2011.
- [8] Cheolwan Kim, Jindeog Chung, "Aerodynamic Analysis of Tilt-Rotor Unmanned Aerial Vehicle with Computational Fluid Dynamics", Journal of Mechanical Science and Technology (KSME Int. J.), Vol. 20, No.4, pp. 561-568, 2006.

★ ★ ★