International Journal of Engineering & Technology, 7 (4.25) (2018) 18-23

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper

Contra Rotating Ducted Propeller Analysis using Computational Fluid Dynamics

Putra Adnan Fadilah*[1], Dadang Furqon Erawan[2]

Chroma International Bandung – Indonesia | www.chromaintegrated.com
**Corresponding author E-mail: putra@chromaintegrated.com*

Abstract

Propeller is well known as a mechanical device for propelling a boat or aircraft, consisting of a revolving shaft with two or more broad angled blades attached to it. For now propeller is used in many types of aircraft and boats. This paper is focusing on the propeller, which is applied for small aircraft such as Unmanned Aerial Vehicles (UAV) where the size of the vehicle can be one of the design constraint. Tandem propellers configuration can be an option to cope with the size constraint. This paper studies the performance of tandem propellers in contra-rotating configuration. The study begins with the validation of Computational Fluid Dynamics (CFD) calculation by comparing the results in term of thrust coefficient to the existing performance data. The case study used an UIUC standard 10 inch propeller. The study of contra rotating ducting propeller is performed by analysing the single ducted propeller and followed by the tandem contra rotation ducted propellers set at different axial distances. The results show that the single ducted propeller performs better than open propeller, and there is an optimal distance between propellers in tandem configuration which gives maximal thrust coefficient.

Keywords: *Contra rotating propeller, ducted, CFD, thrust, efficiency*

1. Introduction

A Propeller is a type of fan that transmits power by converting rotational motion into thrust. A pressure difference is produced between the forward and rear surfaces of the airfoil-shaped blade, and the fluid (such as air or water) is accelerated behind the blade. The analysis of the propeller in this papers is focused on the propeller which is applied to small and medium Unmanned Aerial Vehicles (UAV). The study of the ducted contra rotating propeller is conducted to assess the change in performance characteristic of single propeller when configured in tandem (with contra rotating motion) and installed within a duct. This kind of propeller assembly is interesting to be analyzed since with this configuration the UAV dimension could be reduced. The duct installation in the configuration is purposed for safety reason. Refer to another research from [Ref. 1] Ahmed Farid N., Mohamed Yehia Z. and Aly E. titled Contra-rotating Ducted Fan Aerothermodynamic Design Procedure for Unmanned Application, it is stated that the distance of each rotor can affect the performance of contra rotating assembly blade. Refer to [Ref. 6] ntrs.nasa.gov-Technology and Benefit of Aercraft Counter Rotation Propeller, it is shown that the aircraft with contra rotating propeller (CRP) has higher efficiency than the aircraft with single propeller (SR) only. Another research from [Ref. 2] J.S. Vanderover and K.D. Visser also concluded similarly.
This study will analys the effect of gap between two contra rotating propellers in a ducted configuration.

2. Methodology

2.1. Overview

The methodology which is used in this study is mostly based on Computational Fluid Dynamics (CFD) simulation to predict each propeller configuration performance. Computational fluid dynamic provides a qualitative and to some extent, quantitative prediction of fluid flow by mean of : mathematical modeling (partial differential equations), numerical methods (discretization and solution techniques) and software tools (solver, pre- and post processing utilities).

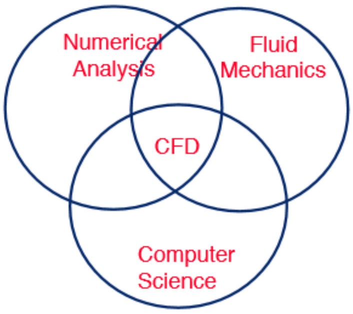


Fig. 1: CFD is interdisciplinary topics

The methodology of this study can be described as following:

- Blade geometry selection from UIUC propeller data sheet/ data base
- Validation of CFD calculation for the selected propeller by comparing the result with UIUC data

- Perform CFD calculation of the above single propeller in ducted configuration and use its result as baseline
- Perform CFD calculation for different axial gap of contra-rotating propeller in ducted configuration to check whether an optimal value exists

For the purpose of comparison analysis, two parameters will be studied, which are CT (Coefficient of Thrust) and J (advance ratio). The definition of those parameters are shown at the following formulaes:

$$C_T = \frac{T}{\rho n^2 D^4} \tag{1}$$

$$J = \frac{V}{nD} \tag{2}$$

The CFD software which is used in this study is NUMECA™ which consists of HEXPRESS for generating unstructured mesh, Autogrid-5 for generating structured mesh, FINE/Open and FINE/Turbo as solver to obtain the simulation result. The post processing use CFView

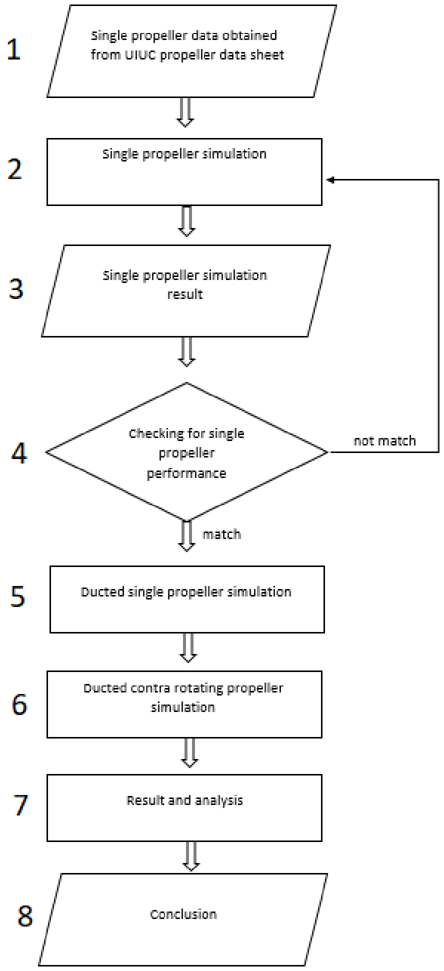


Fig. 2: Flowchart of Methodology

At the step 4 of the above flowchart, the validation of CFD calculation results is conducted by comparing them with propeller performance obtained from UIUC propeller data sheet. Once the CFD result is validated, then other simulation are performed by adding duct and using contra rotating configuration. The performance assessment of the single and contra rotating ducted configuration are conducted by comparing with open single propeller configuration.

2.2. Geometry of the Propeller

The selected propeller for this study is a standard UIUC propeller with 10 inch / ~25 cm in diameter. The propeller geometry is described in the Figure 3:

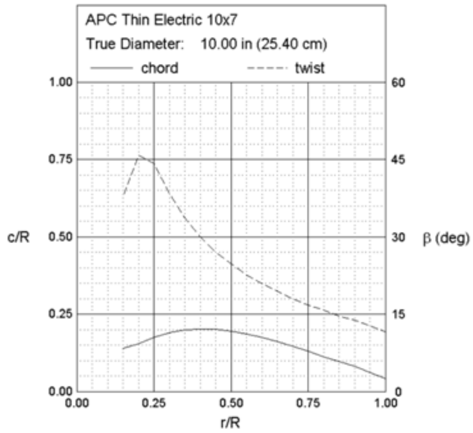


Fig. 3: Single propeller geometry: APC thin electric 10x7

2.3 Cases to be Analyzed

As described previously in the methodology, there will be 3(three) main configurations to be analysed for this study, i.e: isolated single propeller, single ducted propeller and contra rotating ducted propellers. In the last configuration, the effect of axial gap to propeller performance will be analysed. Following Table 1 summarizes each cases that will be discussed in this study.

Table 1: Cases to be analyzed

Case#	Configuration	Remarks
Case 1	Propeller only – Isolated Propeller domain	Comparison to existing performance data
Case 2	Single ducted Propeller	Base line case
Case 3	Contra rotating and ducted	Performance study
#3.1	Tandem with **20** mm axial gap	Study the effect of axial gap to propeller performance in term of thrust coeeficient
#3.2	Tandem with **40** mm axial gap	
#3.3	Tandem with **60** mm axial gap	
#3.4	Tandem with **80** mm axial gap	
#3.5	Tandem with **100** mm axial gap	

2.4. Computational Domain

The computational domain of each configuration propeller is automatically generated by Autogrid-5 or Hexpress softwares. The domain of the simulation is divided into two type of domain, rotating domain which contains the propeller geometry and stationary domain to model the external flow condition. The two different type of the computational domain is connected with interface boundary condition.

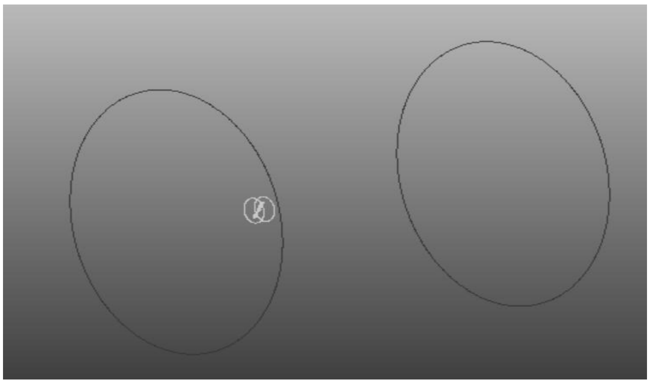


Fig. 4: Isolated propeller domain to modele the external flow over the airfoil

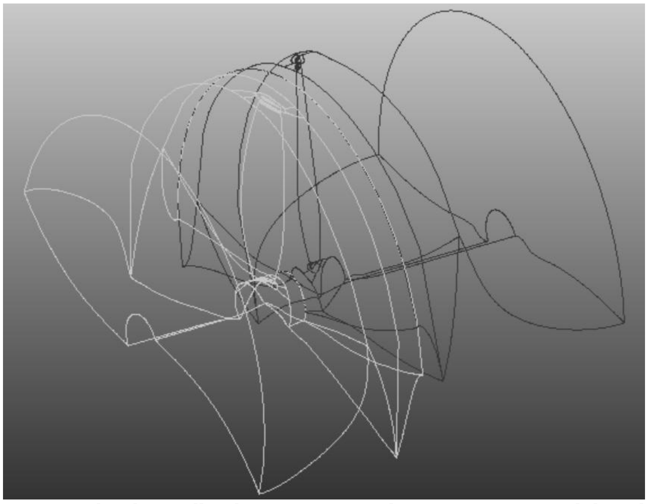


Fig. 5: Isolated propeller domain to assess the contra rotating and ducted configuration

2.5. Meshing

The meshing that is applied on the geometry of each cases use two type of meshing, structured mesh and unstructured mesh (cartesian meshing). For isolated propeller analysis with external flow (Case #1 and #2) HEXPRESS mesher software is used to generate mesh over the propeller and external domain. HEXPRESS is unstructured mesher with cartesian mesh type. For tandem configuration, the structured mesh is used. The contra rotating domain is generated with Autogrid-5 which is a mesher software to generated mesh over turbomachinary geometry especially with shroud and hub configuration. The output of this software is structured mesh with good quality mesh even using default setting.

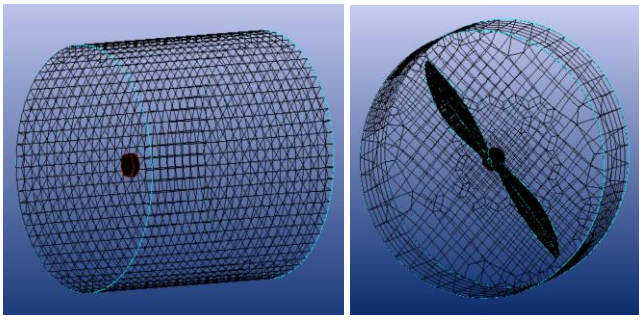


Fig. 6: Mesh that is applied on the external (stationary domain) and rotating domain of the propeller

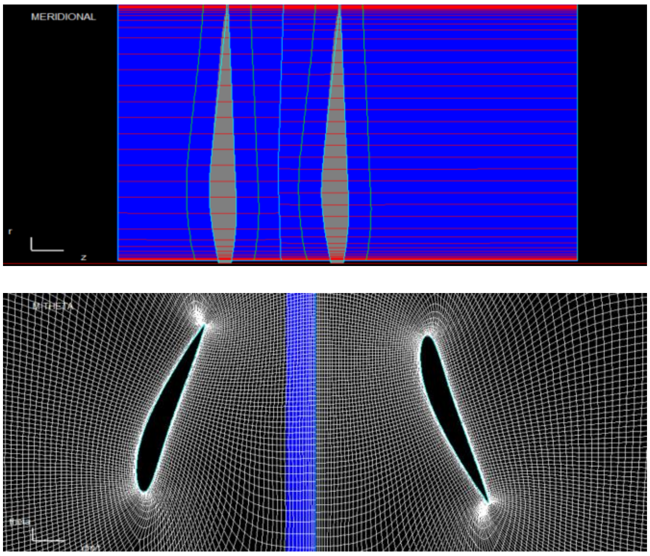


Fig. 7: Structured mesh (2D) of the contra rotating domain

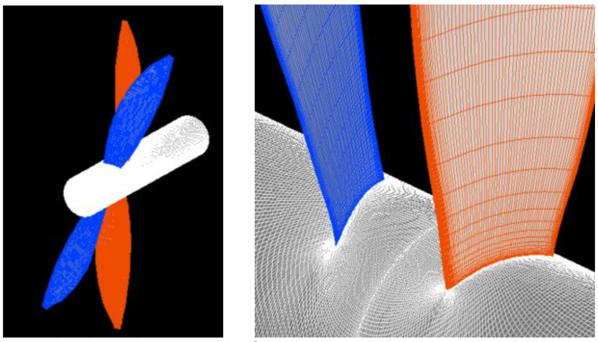


Fig. 8: the three dimensioanl of structured mesh contra rotating propeller

3. Simulation Setting

The simulation setting for CFD calculation consist of defining the inlet mass flow/velocity, external flow condition for the isolated propeller simulation, setting the outlet condition and the rpm of each propeller.

3.1. Boundary Condition

Boundary condition of this simulation is mostly similiar for each configuration (for isolated propeller simulation and ducted contra rotating propeller simulation).

Inlet condition	: mass flow and velocity setting
External condition	: free stream velocity
Outlet condition	: Average static condition
RPM	: given

2.2. Numerical Model

The equation that is used in this simulation is Turbulent navier-stokes, with Spalart Allmaras turbulence model. The number of mesh that is applied on the computational domain geometry is around 4 million to 5 million meshes. The type of the simulation is frozon rotor simulation, this simulation type uses relative motion of the flow in the domain to simulate the rotational motion of the rotor blade. In other world, the rotor remains but the flow in the rotor domain moves at given RPM.

3.3. Convergence Criteria

As criteria of convergence in the simulation, residual growth of mass and momentum is used, as shown in Figure 9. The residual is one of the most fundamental measures of an iterative solutions convergence, as it directly quantifies the error in the solution of the system equation. In a CFD analysis, the residual measures the local imbalance of a conserved variable in each control volume. For complicated problems, however, it is not always possible to achieve residual levels as low as -6 (global residual scale in NUMECA). However, if the global residual curve cannot reach certain low level, the convergence of the simulation still can be assessed by monitoring integrated quantities such as force, drag, mass flow or average temperature. When the simulation converges the integrated quantities value are not too much fluctuating any more. Figure 10 shows the convergence history of integrated quantities that is represented by mass flow.

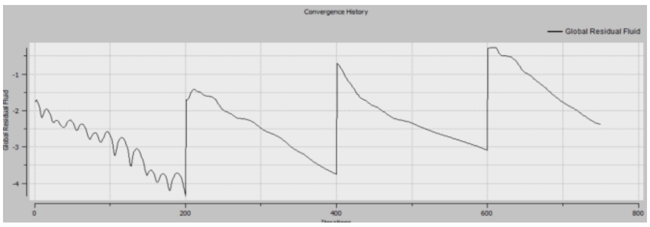


Fig. 9: Residual convergence hystory of the simulation

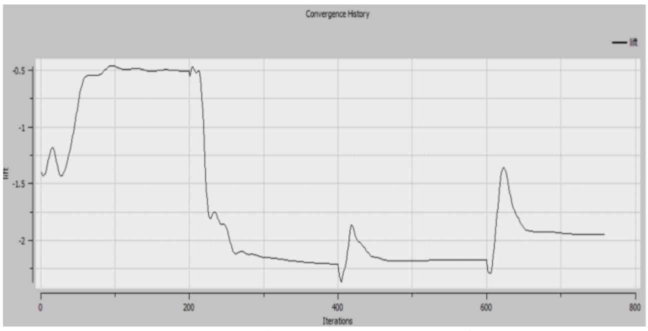


Fig. 10: Integrated quatities convergence hystory.

4. Simulation Result

In this section will be discussed the simulation results. Propeller performance curves will be compared to assess the effect when single propeller is stacked with contra rotating movement and added with ducting.

4.1. Isolated Propeller Domain, Case #1

Table 2: Simulation result of isolated propeller domain for several flow conditions

Condition	RPM	V (m/s)	Thrust (N)	J	CT
1	4000	10	1.107	0.6	0.052
2	4000	7	1.94	0.4	0.091
3	4000	5	2.382	0.3	0.112
4	4000	3	2.683	0.2	0.126
5	4000	1	2.178	0.06	0.102

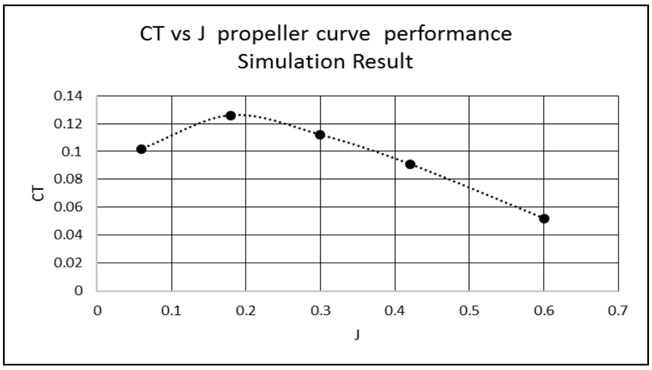


Fig. 11: CT (Thruts Coefficiet) versus J (advance ratio) based on the isolated propeller CFD

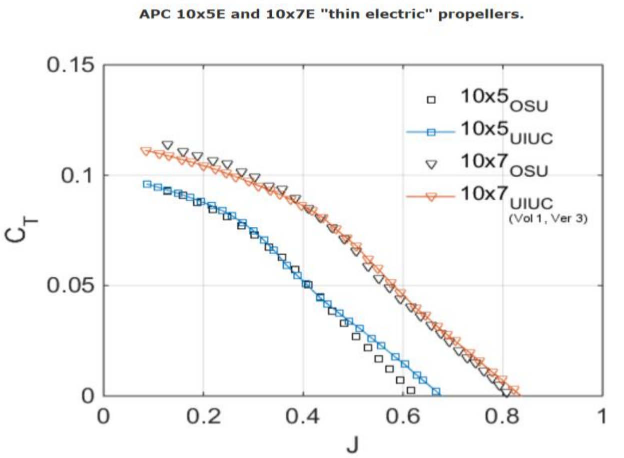


Fig. 12: UIUC propeller data for 10x5 and 10x7 thin electric propeller

The CFD results of the single propeller in of the Case#1 shows that its CT vs J curve has a trend lying between the characteristic of 10x7 UIUC and 10x5 UIUC thin electric propellers. Based on this comparaison it is hence concluded that the CFD setting and calculation result are acceptable to be used further in this study.

4.2. Single Ducted Propeller Simulation, Case #2

In this section will be discussed the simulation result of single ducted propeller configuration. The simulation is conducted to calculate the propeller within duct configuration. The geometry of the duct as shown in the Figure 13 is a simple duct with 1 mm thickness and rounded at the front section.

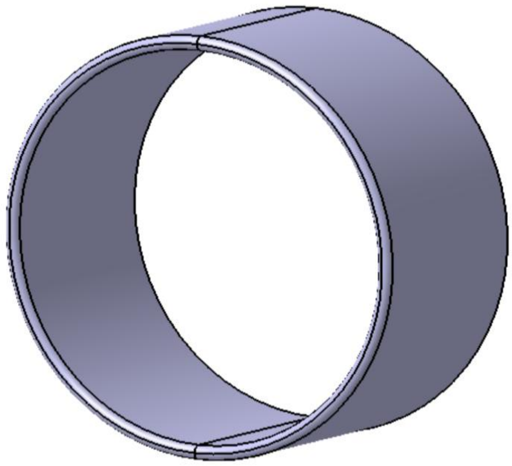


Fig. 13: The duct geometri that modeled to be joined with the propeller

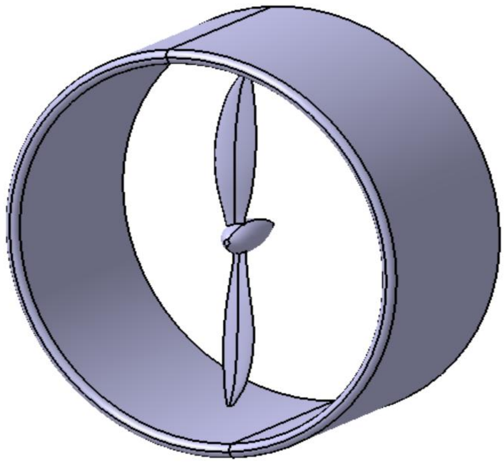


Fig. 14: Computational model of ducted propeller (duct and propeller geometry) assembly

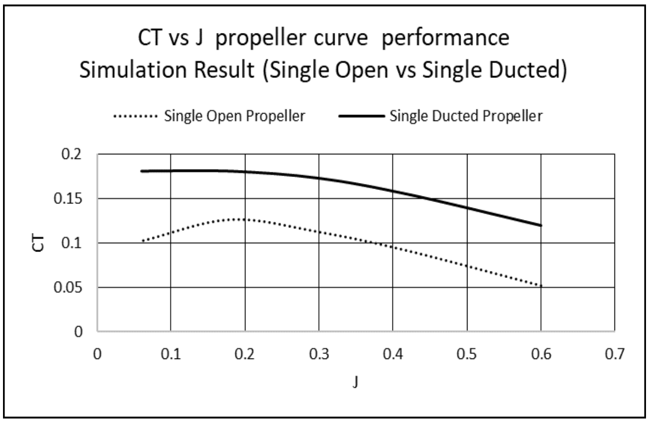


Fig. 15: Comparison CT vs J curve between single open propeller and single ducted propeller

The results compared in Figure 15, shows that the ducted propeller has higher CT (Trust Coefficient) than non ducted propeller.
This case can be used as baseline for further study about tandem ducted propellers

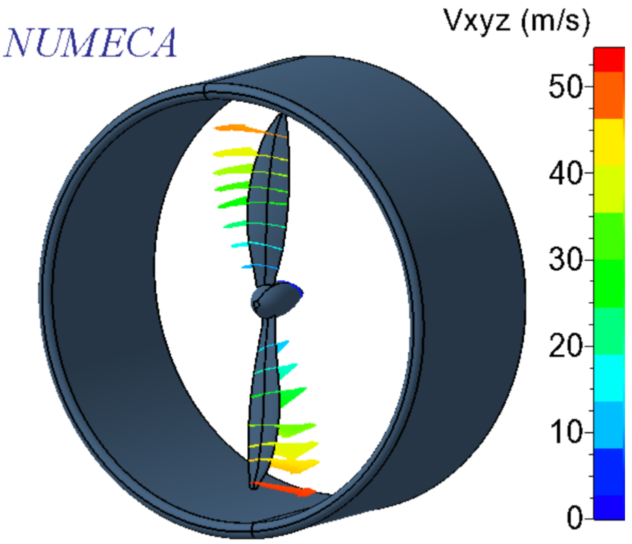


Fig. 16: Velocity distribution along the spanwise propeller

4.3. Contra Rotating Configuration, Case #3

In this section will be discussed the contra rotating ducted propeller configuration. In this configuration as explained previously, the same propellers are stacked within the above duct with contra rotation movement at the same RPM. The analysis is conducted to study the effect of the distance between propellers in tandem configuration to the performance of the assembly in term of thrust coefficient. The gap between propeller 1 and propeller 2 varies between 20 mm (Case #3.1) and 100 mm (Case # 3.5). Since the focus of interest in this section concerns the effect of propellers gap, therefore the computational domain only models the duct and the propeller. The meshing applied on the contra rotating domain is structured mesh generated by Autogrid-5 as shown previously in the Figure 8.

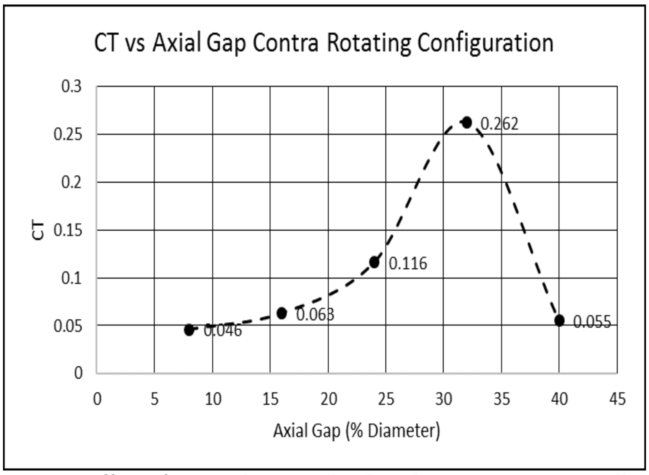


Fig. 17: Effect of an axial gap to the ducted contra rotating propeller coefficient of thrust (CT), at J = 0.4

The results shown in Figure 17 shows that there is an optimal gap which gives the maximal thrust of the tandem assembly propeler. The highest thrust is reached with axial gap of 80 mm which represents 32% of propeller diameter. In the Figure 18, it is shown the comparison of CT vs J curves of three studied configurations: isolated propeller, ducted propeller and contra rotating ducted propeller with optimal gap (32% D). It shows the significant increase of CT of in the case of contra rotating ducted propeller compared to others case.

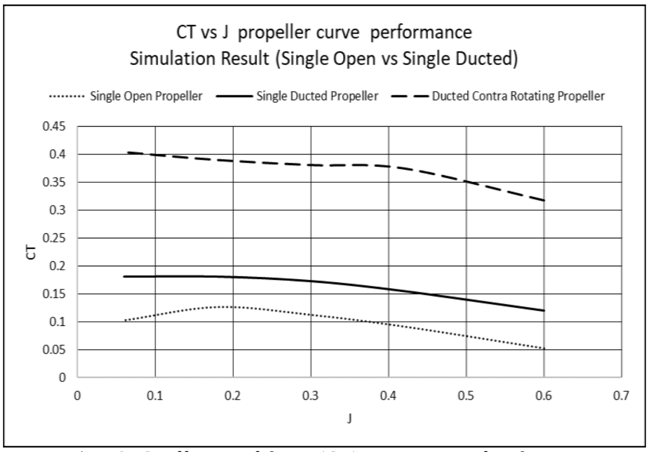


Fig. 18: Coefficient of thrust (CT) comparison of each cases

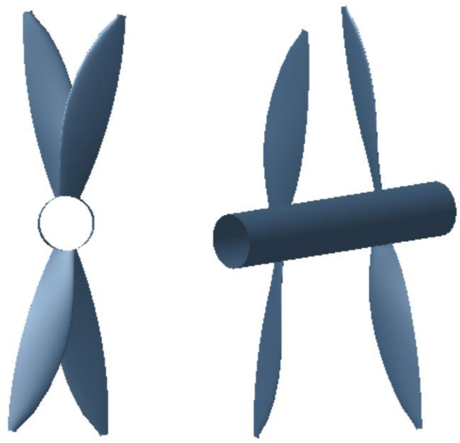


Fig. 19: Geomtery of the contra rotating propeller, the same propeller is just stacking without any modification in pitch blade.

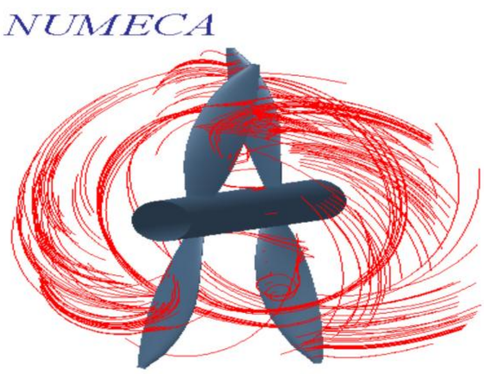


Fig. 20: The streamline that shown the swirlfow and a little of circulation flow when the stream through to the rotation of contra-propeller in the duct.

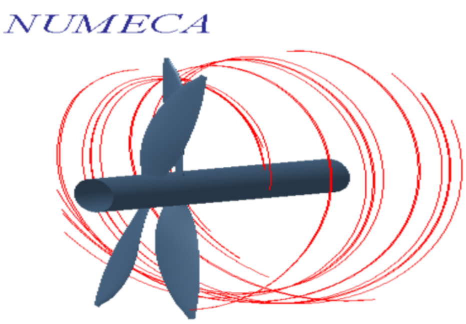


Fig. 21: Flow visualization of tip swirl flow at the ducted contra rotating (isometric view)

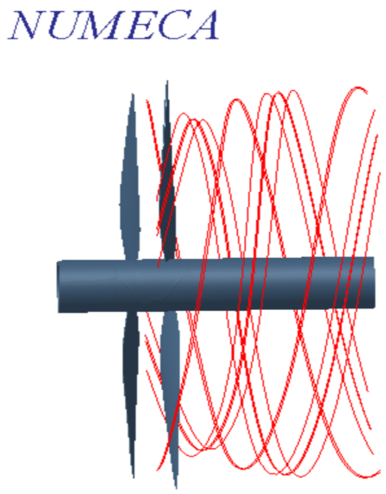


Fig. 22: Flow visualization of tip swirl flow at the ducted contra rotating (Side view)

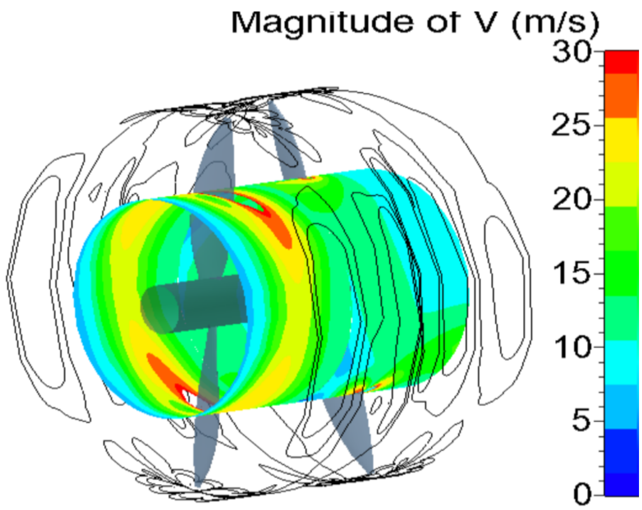
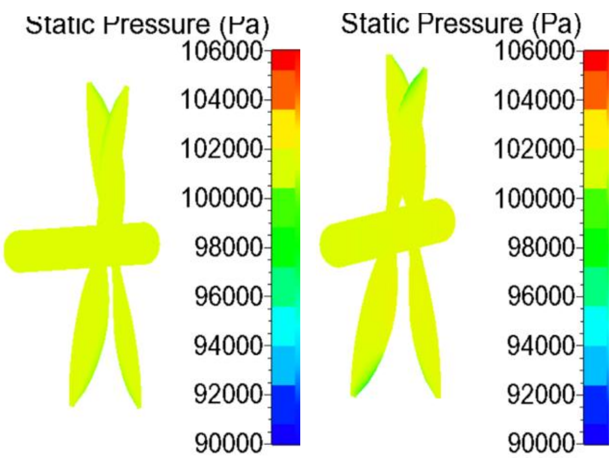
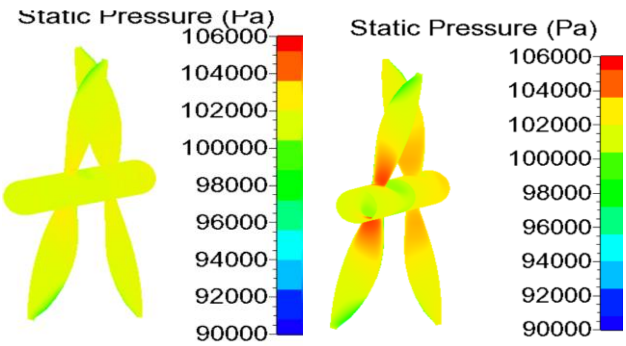


Fig. 23: Velocity contour of the contra rotating propeller domain shows the velocity change through the duct.

Figure 19 to Figure 23 show the simulation result as qualitative form with streamline and color contour. In The Figure 20 which depicts the stream line over the contra rotating configuration is clearly shown the swirl of the flow through propeller, also in the Figure 21 and 22 are shown the tip swirl flow in the ducted contra rotaing domain. There is a little bit recirculation flow at the near propeller wall. Figure 23 present the color contour of velocity in the contra rotating duct.



(a) (b)



(c) (d)

Fig. 24: Pressure contour taken from rear view of contra rotating configuration, Figure (a) shows Case #3.1, (b) Case# 3.2, (c) Case #3.3 (d) Case #3.4.

In conformity to the result shown in the Figure 17 and 18, the higher trust provided by ducted contra rotating propeller has also higher pressure as shown in Figure 24 (d) which is the case of optimal gap.

5. Conclusion

In this study of contra rotating ducted propeler, several cases have been analyzed, starting from single open propeller in free stream flow, followed by single ducted propeller, and finally contra rotating ducted propeller. The first case was intended to check the CFD calculation result by comparing to the existing data. It was concluded that the performed CFD simulations are acceptable to be used for this study since it give a correct trend of CT vs J curve. The next analysis was to perform a CFD calculation of a single ducted propeller which will be used as the baseline for the study. Afterward, the main study is performed on the performance of a tandem contra rotating propeller within ducted configuration which mainly to study the effect of gap between two tandem propeller in the mentioned configuration. It is then concluded:

- the ducted configuration provide better performance than open configuration for a single propeller
- there is an optimal distance between propeller in tandem configuration which gives the best performance in term of thrust coefficient

This study shows that the use of tandem contra rotating propeller within duct is a prospective solution for drone/ UAV application to obtain bigger thrust within a given foot print.

References

[1] Ahmed Farid Nemnem, Mohammed Yehia Zakaria, Aly Elzahaby, “Contra-rotating Ducted Fan Aerothermodynamic Design Procedure for Unmanned Applications”, *2018 AIAA SciTech Forum, 10/2514/6.2018-0745*, 2018.

[2] J.S. Vanderover and K.D. Visser, *Analysis of a Contra-Rotating Propeller Driven Transport Aircraft*, Clarkson University, Postsdam, New York, 13699-5725.

[3] Q.H. Napurwala, *Ducted Fans and Propellers*, M.S. Ramaiah School of Advanced Studies, Bengalore, PEMP RMD 2501.

[4] D. S. Douglas, S. Mills, E. Kisielowski, *Comparative Performance Study of Propeller, Ducted Propeller, and Turbofan Propulsion System*, U.S. Army Aviation Materiel Laboratories, Fort Eustis, Virginia, 1968.

[5] Maria Grazia De Giorgi, Teresa Donateo, Antonio Ficarella, Donato Fontaanarosa, Anna Eva Morabito, and Luca Scalinci, “Numerical Investigation of the Performance of Contra-Rotating Propeller for a Remotely Piloted Aerial Vehicle”, *72nd Conference of the Italian Thermal Machines Engineering Association*, ATI2017.

[6] Technology and Benefit of Aercraft Counter Rotation Propeller. ntrs.nasa.gov

[7] UIUC propeller data base