

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 4, April 2018

Numerical Analysis of Propeller Blade Temperature for Optimised Stub of Pusher Configured Turboprop Aircraft

Mahesh S. Mali¹, Vinay C A.², Sanjay V. Kadam³

P.G. Student, Department of Mechanical Engineering, Rajarambapu Institute of Technology, Rajaramnagar, India¹

Scientist, C-CADD, CSIR-NAL, Belur Campus, Bangalore, India²

Professor, Department of Mechanical Engineering, Rajarambapu Institute of Technology, Rajaramnagar, India³

ABSTRACT: The pusher propellers are mounted at the aft end of fuselage, because of this position the exhaust gas is come in contact with the propeller blades. Therefore temperatures of propeller blades are increased. This problem has to be solved without any major engine modifications. This work addresses, CFD Analysis of optimised engine exhaust stub which was designed by CSIR-NAL, to analyse aft mounted propeller blade temperature using RANS based Realizable K-epsilon model. This analysis was done by using commercial software ANSYS Fluent. The special importance was given for generating a mesh of good quality with a finer prism layer along the wall and also keeping a higher density mesh at critical zones. The simulation was done for both current stub and newly designed stub. The results obtained for current stub simulations were validated with experimental data that has been carried out at CSIR-NAL. The results from numerical analysis for both stub shows the temperature change with respect to particular locations and found satisfactory.

KEYWORDS:CFD, Propeller, Stub, Surface Temperature.

I. INTRODUCTION

In light transport aircraft the propeller of pusher configuration is widely used, because of its number of advantages in terms of weight, cost and aerodynamic considerations. For example, the installation of pusher propeller rather than the traction propeller at the aft end of fuselage avoids the interaction of the propeller slipstream with the wing and offers the potential to enable significant portions of laminar flow to be achieved. One of the best advantage of pusher propeller is de-icing boots are not required because in this design the exhaust hot gas is come in contact with the propeller blades. In Pusher type turbo-prop engine, it is important that the interaction of propeller to exhaust gas is minimum and also the exhaust gas not connect itself to the aft of the nacelle body. This might result in overheating of spinner/hub. However, to overcome this problem, some parts are designed without major engine modifications. In this study the exhaust nozzle called as exhaust stub was designed in such a way that exhaust gas flow was diverted. In this design consideration the exhaust stub is trimmed at an angle of 30 degree at an outlet side as shown in Figure 1.

The Experimental study is very expensive and time consuming, so the present study deals with computational analysis by using the commercial software ANSYS Fluent [1]. The general goals were to analyse the temperature on propeller surface for existing stub and designed stub at given operating conditions. The computational simulations were carried out at CSIR-NAL HPC facility for Solution and post processing. For this analysis, HP Z840 four processor workstation was used.

Notable researchers have worked on the experimental and numerical performance evaluation of pusher type propeller system. The literature survey was carried out to understand the model studies, discretization methodology, working

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 4, April 2018

principle, etc. which was relevant to the problem statement. PremKumar PS et al. [3] carried out experimental and numerical studies on the optimization of oil cooler duct position for a pusher type LTA. In numerical analysis propeller flow field was modeled for the purpose of estimating the thrust loss. The computational analysis was done in Ansys CFX by using $K\omega$ -SST Turbulence model. From the computational results, they conclude that the thrust loss due to the placement of oil cooler at the upstream of the propeller is within acceptable limits. Manish Sharma et al. [5] studied the viscous, incompressible and steady state flow over an F-16 aircraft using CFD tools. In this study the unstructured mesh was created by using ICEM CFD. Fine mesh was formed at critical areas. For CFD analysis K-epsilon model was used. The obtained results are compared with the experimental results and were quite co-related and comparable. S. Subhas et al. [7] also carried the Numerical analysis for marine propeller to study about the flow and cavitation. The meshing was done by creating structured grid with hexahedral cells. In this the study was carried out for two conditions that for propeller at cavitation and propeller at non-cavitation condition. After analysis they conclude that Computation results are in good agreement with experimental findings.

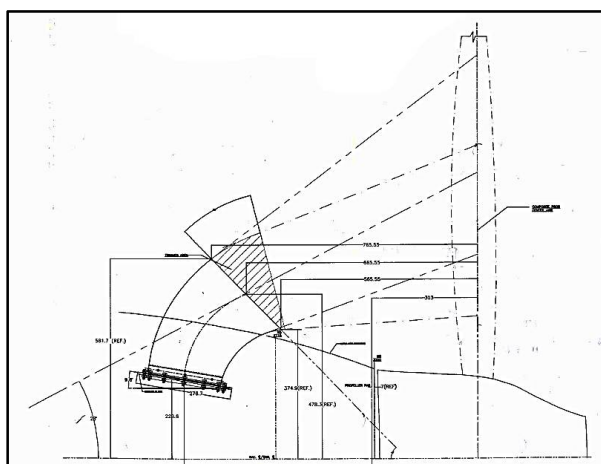


Fig. 1. Stub Layout

II. METHODOLOGY

1.1 MODELLING

For capturing the actual geometry, accurate geometric modelling is required. The model was made as simple as possible for reducing the computational time and meshing efforts. Therefore it was decided to neglect insignificant and inconsequential structural details. Also, the fuselage and nacelle inside parts were not considered and nacelle intake lip was closed as shown in Figure 2. The geometric modelling was carried out using CATIA V5 R20. The geometry involves stationary and rotating parts. The stationary part consists of nacelle and exhaust stub. The rotating part consists of propeller with spinner as shown in Figure 2.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 4, April 2018

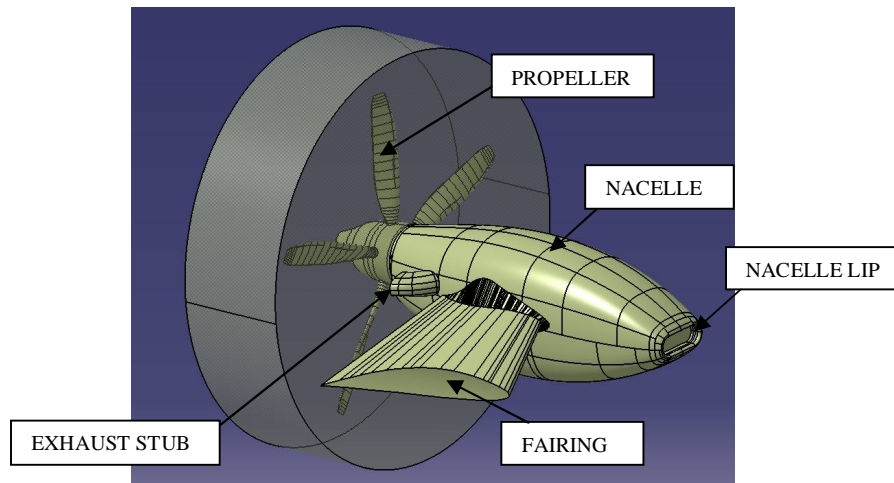


Fig. 2. Model with part names

The cylindrical shape domain was selected for this study for minimum distortion [2]. It was generated in CATIA V5 R20. The full domain is shown in Figure 3. X and D are the length and diameter of the model. To capture the rotation of propeller, cylindrical domain was created which contains the rotating parts as shown in Figure 2. [3]

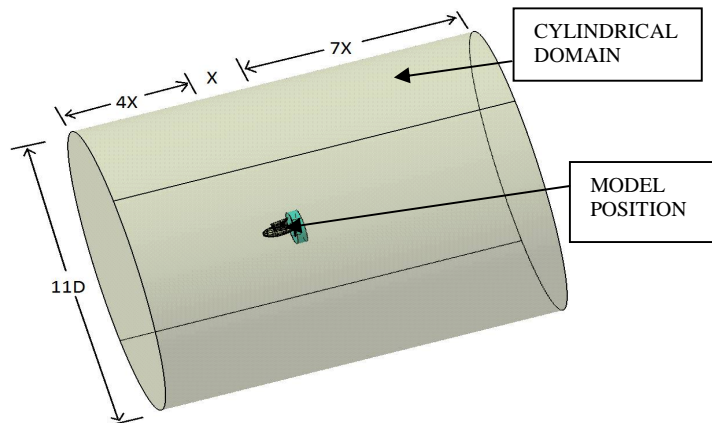


Fig. 3. Model with part names

1.2 MESH GENERATION

The modelled geometry was saved in IGES format and imported to ANSYS ICEM CFD. The unstructured (Tetrahedral) mesh was generated using patch independent octree method [4] [6]. Small element size was given to capture the critical areas. To capture boundary layer, fine prism layers were generated along the wall. Smoothing of elements was carried out to improve the quality of mesh. After completion of mesh 20.5 million cells were found. Figure 4 and 5 shows the mesh model with major boundaries.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 4, April 2018

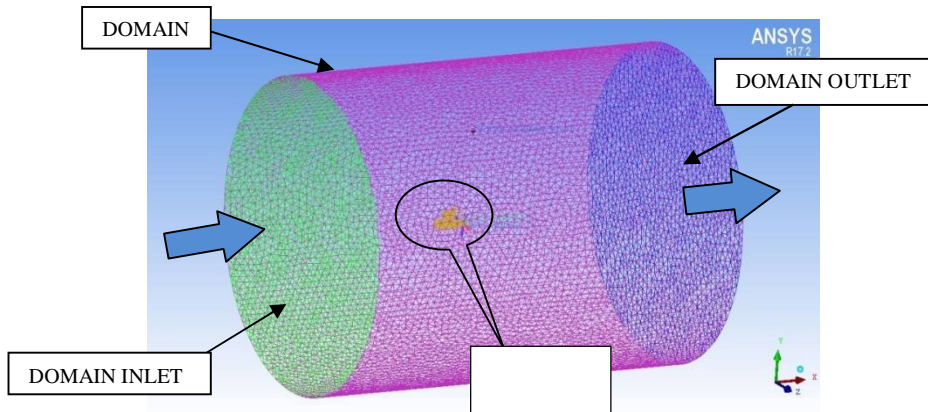


Fig. 4. Mesh Model with Global BC

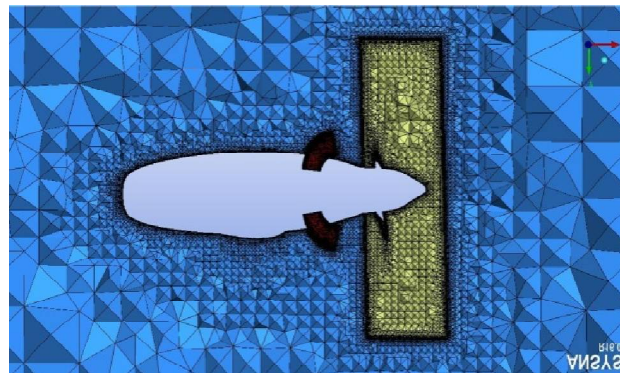


Fig. 5. Cut section of mesh model

1.3 CFD ANALYSIS

For this computational analysis ANSYS FLUENT 16.0 was used. In this, pressure based solver was selected with double precision. The Realizable k- epsilon model was used as turbulence model for this analysis. This is a two equation RANS model which predicts the flow separation from smooth surfaces. The working fluid is assigned as air for domain flow fluid and carbon di-oxide for exhaust fluid [5]. Both are assumed to be incompressible. The SIMPLE algorithm was selected for pressure velocity coupling. The appropriate boundary types and domain interfaces were applied to this computational model as represented in Table 1. To control the solution the under relaxation factors were monitored. This analysis was done for ground static condition. The operating conditions of this flight condition are shown in Table 2.

Table 1. Boundary conditions

Sr. No.	Part name	Boundary settings	Magnitude	Temperature
1	Domain inlet	Velocity inlet	6.8 m/s	307 K
2	Domain outlet	Pressure outlet	-	-
3	Domain Wall	Pressure outlet	-	-

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 4, April 2018

4	Nacelle	Wall	No slip	-
5	Stub inlet	Mass flow inlet	0.5935 m/s	913 K
6	Stub outlet	Interior	-	-
7	Stub wall	Wall	No slip	-
8	Spinner	Wall	No slip	-
9	Propeller Blades	Wall	No slip	-
10	Rotational domain	Interior	-	-

Table 2. Operating condition

Condition	Altitude, m (ft)	OAT, °C (K)	Speed m/s (Mach no.)
Ground Static Condition	914.4 (3000)	34 (307)	6.8 (0.02)

III. RESULTS AND DISCUSSION

After setting boundary conditions, the setup was run with convergence criteria of 10^{-5} . The solution was carried out for nearly 15000 iterations. The simulation takes about 65 CPU hours to achieve this order of convergence. In post processing, less than 0.4 % mass imbalance was found which implies that continuity equation was satisfied.

Figure 6 (a) and (b) represents the contours of velocity magnitude at cut plane of model with current stub and optimised stub respectively. Velocity contours at stub indicates that the exhaust gas coming from stub and mixing with the outer air because of propeller rotation. A high velocity region was observed at downstream side of propeller, it indicates that a greater amount of thrust was developed by propeller.

Figure 7 (a) and (b) shows the velocity contours at propeller plane. These contours give a clear picture that the velocity profile was generated from trailing edge of propeller. Due to the propeller rotating in clockwise direction at 1700 rpm, the velocity profile was generated at the opposite side of rotation, which is of maximum value of 232 m/s near to the blade surface.

Figure 8 and 9 shows the contours of static temperature at upstream and downstream side of propeller. From the temperature contours of current stub it was found that the temperature is high at middle region of propeller. But in optimised stub the temperature is high at tip side of propeller blade. The temperature contours at downstream side of propeller blades shows good and fine temperature profile because when propeller is rotating, the downstream face of propeller blades were come in contact with the exhaust gas.

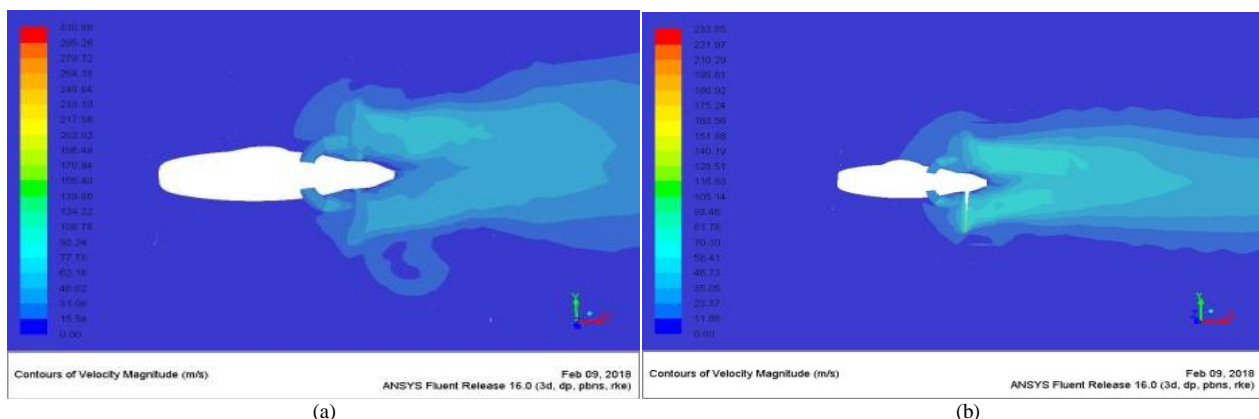


Fig. 6. Velocity contours at cut plane of model (a) for Current stub model, (b) for Optimised stub model

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 4, April 2018

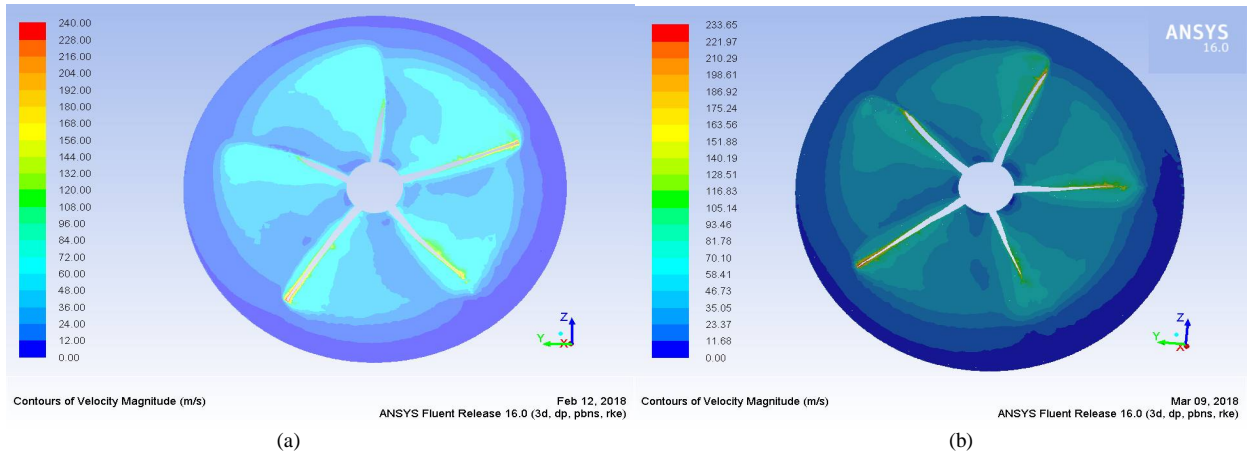


Fig. 7. Velocity contours at propeller plane (a) for Current stub model, (b) for Optimised stub model

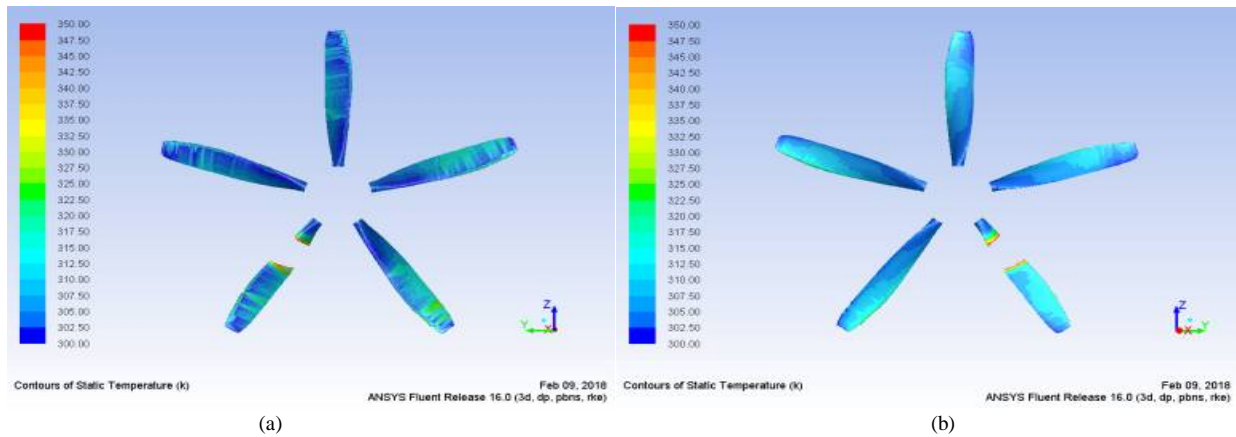


Fig. 8. Temperature contours on (a) upstream side of propeller (b) downstream side of propeller for current stub model

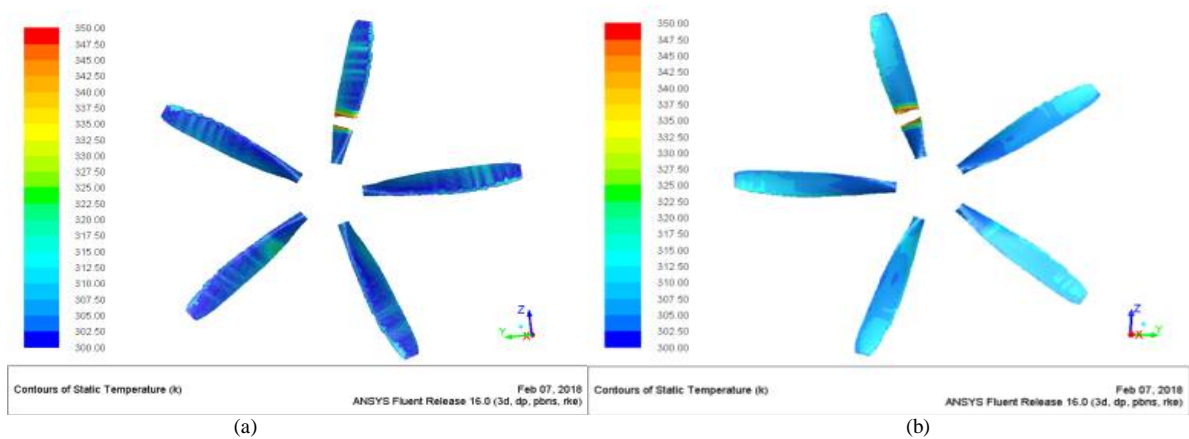


Fig. 9. Temperature contours on (a) upstream side of propeller (b) downstream side of propeller for Optimised stub model

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 4, April 2018

3.1 RESULTS VALIDATION

The experimental analysis has been carried out at CSIR-NAL. In experimental analysis the temperature of second blade (when counted from top to clockwise direction) was examined at camber side of propeller at every 110 mm distance. In CFD post-processing, for current stub model, temperatures of same blade with respect to the particular distance were taken. From this analysis it is observed that the CFD results are in good agreement with experimental results. From this baseline, the next analysis was carried out.

In numerical analysis the temperature results are taken for both current and optimised stub models. For comparison of results graph was plotted as shown in Figure 10.

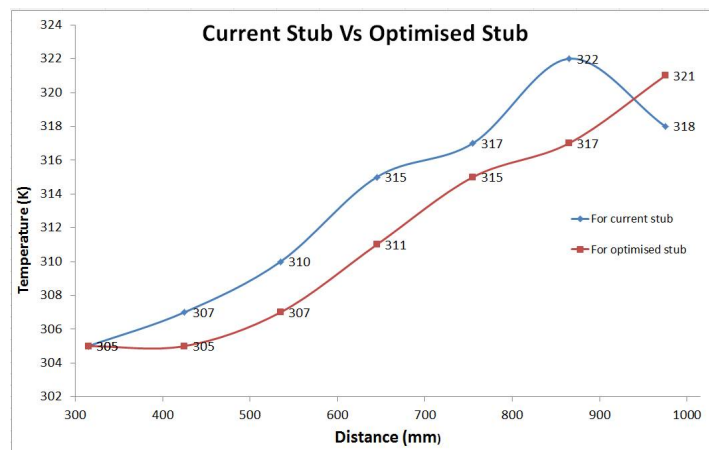


Fig. 10. Comparison of temperatures on propeller blade

Figure 10 indicates the temperature difference with respect to particular distance on propeller blade for current and optimised stub model. From the above plots it is observed that for optimised stub model the temperature was increased from root to tip of the blade, it occurred because of exhaust gas flow was diverted to tip side of propeller blade.

IV. CONCLUSION

In this paper the computational study has been effectively carried out using ANSYS Fluent for nacelle, exhaust stub and propeller of Light transport aircraft for pusher configuration. The simulations were carried out for ground static condition, to determine the temperature distribution on propeller blade surface and velocity across the propeller. The CFD results for optimised stub are promising and confirm that the exhaust gas flow is diverted and the temperature on propeller blade is dropped. From the overall analysis the results obtained for new designed stub was found satisfactory at given operating condition. The same model will be used for further simulations for various flight conditions. These preliminary CFD outcomes have supported to get clearance from certification agencies for directing the ground/flight trials.

REFERENCES

- [1] JiyuanTu, Guan HengYeoh and Chaoqun Liu, "Computational Fluid Dynamics: A Practical Approach", First Edition, USA, 2008.
- [2] Mateusz Stajuda, MaciejKarczewski, Damian Obidowski and Krzysztof Jozwik, "Development of a CFD model for propeller simulation", Mechanics and Mechanical Engineering, Vol. 20-No.4, PP. 579-593, 2016.
- [3] PremkumarPottanamSelvarajan, Senthil P. Kumar C, ElangovanSrinivasan, and BaskarChakravarthy, "Optimising Oil-Cooler Duct Position for a Pusher Type Turboprop Aircraft", Aviation Technology, Integration, and Operations Conference, AIAA AVIATION Forum, (AIAA 2013-4331), Los Angeles, CA, 2013.
- [4] Eric W. M. Roosenboom, Arne Stürmer, Andreas Schröder, "Advanced Experimental and Numerical Validation and Analysis of Propeller Slipstream Flows", Journal Of Aircraft, Vol. 47-No. 1, pp. 284-291,2010.



ISSN(Online): 2319-8753
ISSN (Print): 2347-6710

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 4, April 2018

- [5] Manish Sharma, T. Ratna Reddy, and Ch. Indira Priyadarsini, "Flow Analysis over an F-16 Aircraft Using Computational Fluid Dynamics", International Journal of Emerging Technology and Advanced Engineering, Vol. 3, Issue 5, pp. 339-343, 2013.
- [6] Jianping Yin & Arne Stuermer, "Coupled uRANS and FW-H Analysis of Installed Pusher Propeller Aircraft Configurations", 30th AIAA Aeroacoustics Conference, Miami, Florida, 3332, 2010.
- [7] Jianping Yin & Arne Stuermer, "Aerodynamic and Aeroacoustic Analysis of Installed Pusher-Propeller Aircraft Configurations", Journal of Aircraft, Vol. 49-No. 5, pp. 588-599, 2012.
- [8] S. Subhas, V F Saji, S. Ramakrishna and H. N Das, "CFD Analysis of a Propeller Flow and Cavitation", International Journal of Computer Applications, Vol. 55-No. 16, pp. 26-33, 2012.
- [9] Aviation & Emissions A Primer, Federal Aviation Administration Office of Environment and Energy, 2005.