International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 10, October- 2014

Computational Analysis of Proprotor in Tilt Rotor Configuration

B. Prashanth Sagar Reddy, B.E Aeronautical Engineering (2010-2014)
K.C.G College of Technology,
Dept. of Aeronautical Engineering,
Chennai-97

***Abstract* –Tilt rotor aircraft is a convertible rotor configuration which can perform the roles of both helicopter and fixed wing aircraft. By their design, tilt rotors can cruise significantly faster than helicopters, and so offer at least one major performance advantage. The tilt rotor aircraft is also a compromised engineering design, and it is not aerodynamically efficient as helicopter in hovering or as efficient as an fixed wing aircraft in forward flight. The rotor system of tilt rotor called proprotor which is a combination of propeller and rotor blade design is the major cause for less efficiency of a tilt rotor aircraft. Various methods have been suggested until now to improve the efficiency of proprotor blades. These include the variation of blade twist, blade RPM, blade geometry. This work involves in computational analysis of proprotor blades by observing the performance variation through applying the suggested methods. This performance is measured in terms of force generated per unit area of the blades in both forward and hovering conditions. Blade geometries of ONERA209, TRAM are considered for the analysis. The work is carried out using CFD softwares ANSYS ICEM-CFD, ANSYS CFX.**

Keywords: Tiltrotor, Proprotor, CFD

I INTRODUCTION

CFD is the analysis of fluid flow problems and other related fluid phenomenon in any system, in a computer based simulation. It is an intricate set of codes involving the basic fluid equations- mass, momentum, and energy equations. These codes replace the basic fluid equations that are integral or partial differential form to discretized algebraic form. These algebraic equations are applied to small domains in space or time to obtain numerical solution by an iterative process.

Today CFD is used as a research and design tool. They are used to get data which is approximately analogous to the wind tunnel testing. But in certain cases, wind tunnel testing does not simultaneously simulate many properties that are vital in flight. One such example is testing hypersonic Mach number and high flow field temperatures in trans-atmospheric vehicles.

Pursuing a wind tunnel testing on the same is not encouraged due to many reasons including engineering feasibility and mainly high costs. In such times CFD becomes handy in designing and running numerical analysis on the same.

CFD is coded with numerical algorithms, which are basic fluid flow equations such as Navier-Stokes equation and Euler equations. The codes are coupled with various models that are needed by researchers and other designers in order to simulate the designed physical geometry in the flow field.

The current work utilises the CFD as a tool to analyse the performance characteristics of the proprotor blades. The ANSYS ICEM CFD is a commercial software that starts advanced CAD/geometry readers and repair tools to allow user to progress to a variety of geometry tolerant meshes.

The ANSYS CFX is a post meshing solver and processing tool for the geometry which can be utilised to create domain for the analysis and insert various parameters for the boundary. Furthermore, the results can be obtained after the solver stops the convergence of solution to the analysis.

II ANALYSIS OVERVIEW

The computational analysis in this work is performed in the following steps:

1. Blade geometry is used to create the model for meshing along with the domain dimensions.
2. The domain structure is similar for the two models viz., sphere and cylinder in hovering and forward motion conditions.
3. After the meshing is done, the boundary conditions for the domain such as fluid inlet, outlet and wall are given.
4. Other parameters such as operating conditions (pressure, temperature) are selected along with the turbulence criteria.
5. The results are obtained in the post processing section of the software where the output models can be selected.
6. Calculator that measures force, pressure, temperature etc., at a particular point in the model are also available.
7. With the help these calculator the required measurements of force and area of the blade are obtained.

III MATHEMATICAL MODELLING

In the design environment of ANSYS ICEM-CFD the 3-D design of the computational domain is started with the modelling of the blade geometries of two blades. The rotor configurations are given below.

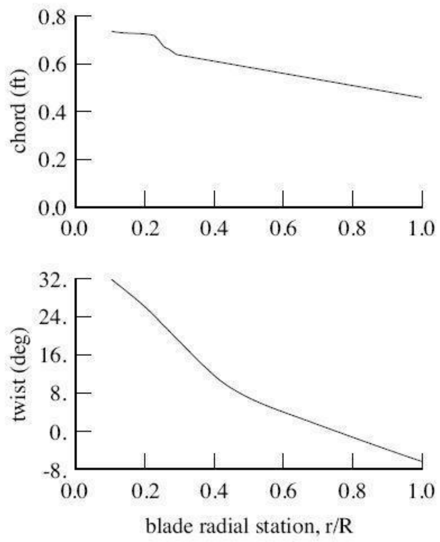
Table 1: configuration of ONERA209 rotor

SL NO	ROTOR ELEMENTS	CONFIGURATION
1	Radius of the rotor disc	5.345 m
2	Number of blades	3
3	Chord length	0.35 m
4	Rotor hub diameter	1.27 m

Table 2: configuration of TRAM rotor

SL NO	ROTOR ELEMENTS	CONFIGURATION
1	Radius of the rotor disc	1.45 m
2	Number of blades	3
3	Airfoil sections	XN28, XN18, XN12, XN09

The twist along the TRAM blade is given by following graph:



Along with the blade modelling, the computational domain or the control volume needs to be modelled. Here, spherical and cylindrical domains are considered for the analysis as to facilitate the two working conditions of the blades viz., hover and forward movement of the aircraft.

Later, the whole domain is subjected to meshing. Unstructured grid is considered for this analysis that uses tetrahedral meshing technique. The following table reveals the information about the domain and the mesh.

Table 3: Domain and Meshing data

Blade/mesh info	OA209	TRAM
Domain volume	43498.8 m^3	1562.68 m^3
Nodes	216506	54374
Elements	111573	273847

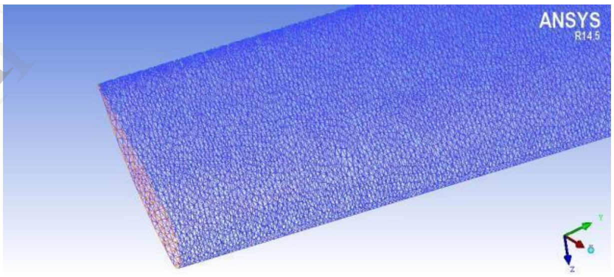


Fig.1 Meshing of blade section

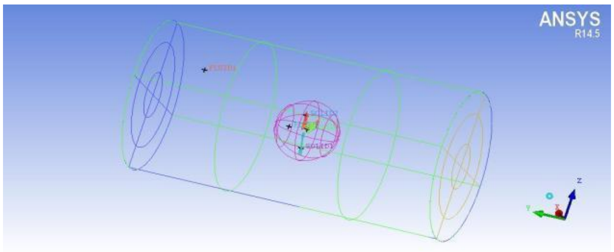


Fig.2 Complete domain before meshing

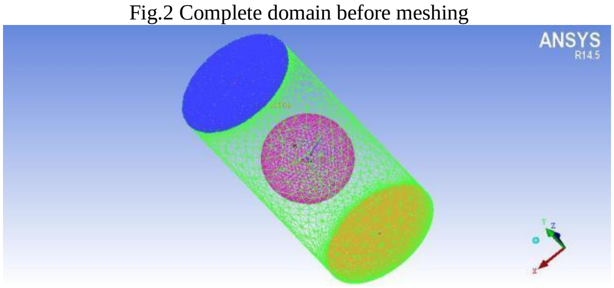


Fig.3: Meshing of total domain

IV BOUNDARY CONDITION AND SIMULATION

The medium inside the domains of sphere and cylinder are defined as fluid whereas the blades are defined as solid medium. Along with the medium, the boundaries are defined as wall, inlet and outlet for the cylinder to simulate the free stream velocity over the blades.

The simulation is performed by considering the following parameters:

Table 4: Operating parameters for model

SNo	Parameters	value
1	Blade RPM	400
2	Temperature	300K
3	Pressure	1 atm

The turbulence model selected is shear stress transport with medium intensity (5%). The blade material is assumed to be Aluminium and the fluid as ideal gas.

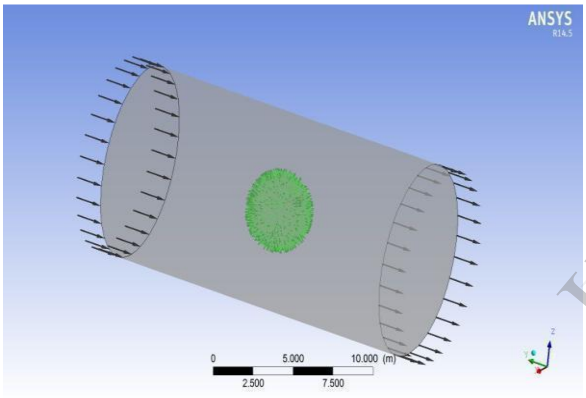


Fig.4 Boundary parameters for model

V RESULTS AND DISCUSSION

By applying the above criteria, results can be obtained in the post processing section of the ANSYS CFX. Various output modes can be selected such as vector plots, streamlines, contours etc., Also, the parameters such as the forces, pressure are obtained for a particular area mentioned.

The results shown are as follows:

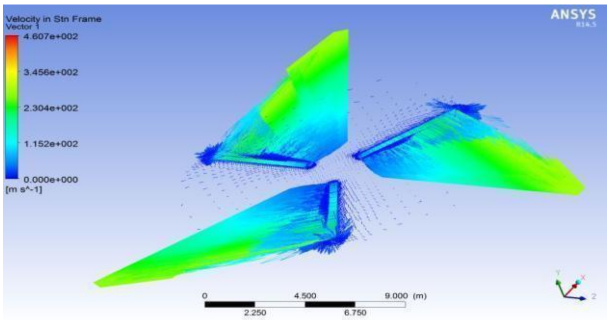


Fig.5 Velocity plot ONERA rotor

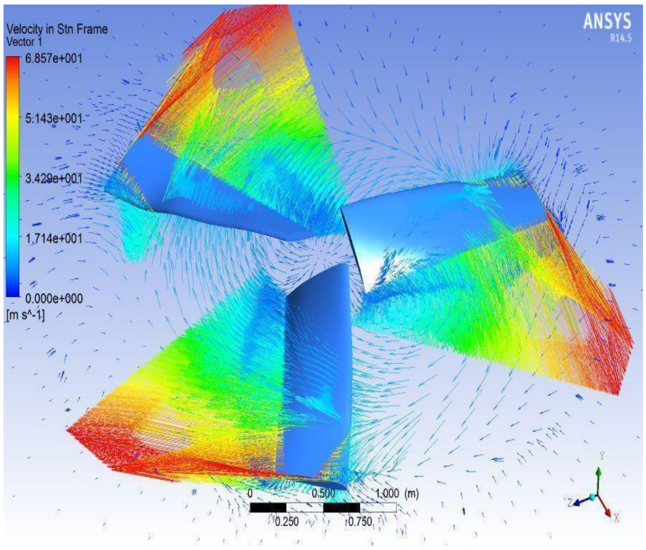


Fig.6 Velocity plot for TRAM blade

From the velocity plots or vector plots the magnitude and direction of the air flow is visualised

Table 5: Results for Hovering condition

BLADE	FORCE GENERATED (N)	AREA OF ROTOR DISC (m^2)	FORCE/AREA (N/m^2)
OA209	3644.22	3.8118	956.03
TRAM	30.46	1.5003	20.30

The results for hovering condition of aircraft (0 degree tilt) are given in the above table. The performance of two rotors can be compared with the values of force/area.

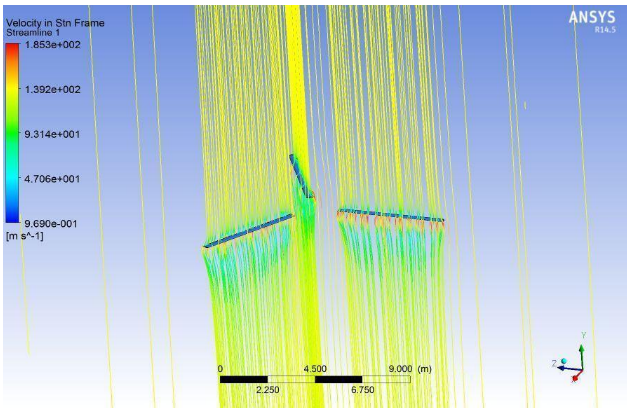


Fig.7 Streamline velocity plot for ONERA rotor

Table6: Results for Full tilt condition of the rotors

BLADE	FORCE GENERATED (N)	AREA OF ROTOR DISC (m^2)	FORCE/AREA (N/m^2)
OA209	171242	3.81	45543.08
TRAM	171422	1.50	114281.33

VI CONCLUSION

The following conclusions can be made from the above results

1) The TRAM blade produces significantly less force in the hovering condition of the proprotor compared to that of ONERA blade.
2) On contrast, the TRAM blade generates more force per unit area along the free stream direction of air than the ONERA blade in forward motion (90 degree tilt) condition.

The reason for this performance variation in two blades is because of the changes in the blade geometry mainly the twist in the blade. Blade twist reduces the area of blade that interacts with the flow which in turn decreases the loss of energy from flow due to interference. Therefore, a twisted blade is more effective when the velocity of motion is high compared to a relatively flat blade. Although the flat blade is effective under hovering conditions that requires more force to lift the aircraft than its motion.

Another reason is the length of blade which is also another cause for loss of energy due to the increase in the surface area of the blade. This analysis clearly verifies the reasons stated that cause the performance variation of blade in tiltrotor configuration. Also, this work emphasizes the need for implementing the methods that are given earlier to improve the performance of proprotor systems in tilt rotors.

VII REFERENCES

1. Wayne Johnson; "Airloads and Wake geometry
2. calculations for an isolated Tiltrotor model in a wind tunnel", 27th European Rotorcraft Forum, Moscow, Russia, September 11-14,2001.
3. Mark A. Potsdam & Roger C. Strawn; "CFD Simulations of Tiltrotor configurations in hover", The American helicopter Society 58th Annual Forum, Montreal, Canada, June 11-13, 2002.
4. E. M. Lee Rausch; "Simulation of an Isolated Tiltrotor in
5. Hover with an Unstructured Overset-Grid RANS Solver", American Helicopter Society 65th Annual Forum, TX, May 27-29, 2009.
6. K. M. Pandey, G.Kumar, D.Das, D.Deka, A.Surana, H.J.Das; "CFD analysis of an isolated main helicopter rotor for a hovering flight", IRACST- Engineering Science and Technology: An International Journal (ESTIJ) Vol. XXX, No.XXX, 2011.
7. Conor William Stalhut; "Aerodynamic design optimization of proprotors for Convertible-Rotor concepts", Thesis- University of Maryland, College Park.
8. phillipe Beaumier, Julien Decours, Thierry Lefebvre
9. Onera; "Aerodynamic and aero-acoustic design of modern tilt-rotor", the ONERA experience; 26th International Congress of the Aeronautical Sciences.
10. Abraham E. Karem; "optimum speed Tiltrotor", US patent US6641365 B2.
11. Ethan Romander; "3-D Navier-Stokes Analysis of Blade Root Aerodynamics for a Tiltrotor Aircraft in Cruise",AHS Vertical Lift Aircraft Design Conference, San Francisco, California, January 18-20, 2006.