

ICMIEE-PI-140413

Simulation investigation on flow characteristics for the flow over a propeller used in VTOL RC aircrafts

S. M. Mahbobur Rahman^{1,*}, Mohammad Mashud¹

¹ Department of Mechanical Engineering,
Khulna University of Engineering & Technology, Khulna-9203, BANGLADESH

ABSTRACT

This paper illustrates the scenario of fluid flow characteristics for the flow over a propeller used in vertical takeoff and landing (VTOL) radio controlled (RC) aircrafts. Simulation investigation has been conducted through SolidWorks flow simulation using a propeller model. Whenever air flows over the propeller, due to the rotational motion a thrust force is generated which will lift the aircraft in the air. This thrust force is important in selecting the motor to be used in the aircraft model. Thrust force along with pressure, velocity, temperature distribution and velocity flow trajectory were determined numerically. These parameters imply the feasibility of using the propeller, consequently selecting the proper power source in order to get a rigid and stable flight of the aircraft.

Keywords: SolidWorks flow simulation, flow characteristics, VTOL RC aircrafts, thrust force, feasibility

1. Introduction

Propellers are the main source of propulsion in general aviation and radio controlled (RC) airplane market. As RC airplanes grow in size, the propulsion systems for these vehicles now become viable sources of propulsion for small drone aircraft which do not have the range and endurance parameters of the predator or Global Hawk systems. The ability to evaluate the performance of these propulsion systems via simulation, as well as their effects on the rest of the flight vehicle, enables the designer to choose engine/blade combinations which are more optimal for the role of the particular vehicle. Accurate simulation of the complete vehicle aerodynamics, including the contributions from propeller flow-field, can lead to robust autopilot design. With these benefits of simulation in mind, now consider the advances and industrial use of CFD over the last decade or so [1]. The flow over a propeller is one of the most challenging problems in the field of computational fluid dynamics (CFD). Due to the airflow over the propeller, thrust force and torque will generate to lift the unmanned aircraft in the air. Various numerical simulation approaches (boundary elements, panel methods, etc.) for studying propeller geometries have been used for decades, but only recently, due to the rapid advances in computer power and in the parallelization capabilities, different CFD methods are increasingly applied to simulate the full three-dimensional viscous and turbulent flow for various propeller geometries [2-5]. In an unmanned VTOL RC aircraft the propeller generates the sufficient thrust to lift the aircraft in the air, which is the prime objective of propeller design and analysis. Through numerical simulation certain thrust forces were determined for different rotational speeds to test the feasibility either the aircraft will fly efficiently or not. The numerical simulation was done in SolidWorks flow simulation and the propeller data were taken from UIUC propeller database site [6].

2. Methodology

The validation of the simulation boundary conditions were initially followed the ASTM standard with Numerical simulation. Flow Simulation analysis was conducted using SolidWorks.

2.1 Propeller Performance and Design Fundamentals

A propeller can be considered as a number of equally spaced advancing rotating blades, kept in rotation by the torque of the engine. The combined lift and drag result in propeller thrust and torque, respectively the axial force and moment. The direct result of the rotation is an increased pressure directly behind the propeller and a decreased in front of it.

2.1.1 Propeller Performance

A propeller blade (Fig.1) is simply a rotating airfoil, similar to an airplane wing, which produces lift and drag. It has both induced upwash and downwash due to the complex helical trailing vortices that it generates. The most two important parameters of a propeller for design and analysis projects such as this are the thrust and torque it produces [7].

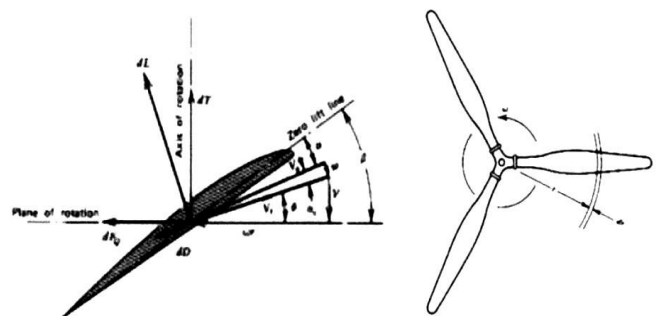


Fig.1 Depiction of a propeller blade cross section

The Thrust of a propeller depends on the volume of air (or water) accelerated per time unit, on the amount of

* Corresponding author. Tel.: +88-01736941309
E-mail address: mahboburr6@gmail.com

the acceleration, and on the density of the medium. Based on momentum considerations, it can be expressed by the following formula:

$$T = \frac{\pi}{4} \cdot D^2 \cdot \left(v + \frac{\Delta v}{2} \right) \cdot \rho \cdot \Delta v$$

T	thrust	[N]
D	propeller diameter	[m]
v	velocity of incoming flow	[m/s]
where: Δv	additional velocity, acceleration by propeller	[m/s]
ρ	density of fluid	[kg/m ³]
	(air: $\rho = 1.225 \text{ kg/m}^3$, water: $\rho = 1000 \text{ kg/m}^3$)	

2.1.2 Propeller Modeling and Selection

In the field of RC aerial vehicles, selection of proper propeller is an important scenario. Depending upon the required flight phenomena, appropriate propeller should be modeled and selected. For this simulation investigation 10*4.7 propeller model was selected because of its appreciable flexibility and light weight (13g approx.). 10*4.7 indicates that the propeller has a diameter of 10 in and pitch of 4.7 in per revolution. Fig.2 represents the design flow-chart for the selected propeller.

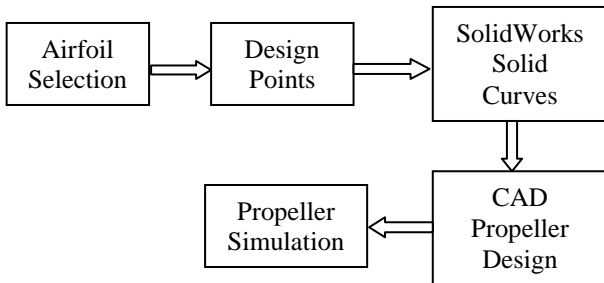


Fig.2 Propeller Design and Analysis Flow-Chart

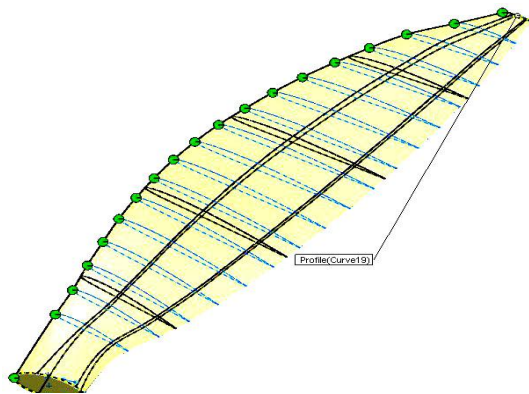


Fig.3 Sketch for loft feature in SolidWorks

Using the data from the UIUC database site [6], the points through x-y-z were generated and then by



Fig.4 Final rendered model of 10*4.7 propeller

Providing a loft feature a solid propeller model was obtained. Fig.3 and 4 illustrates the scenario of generating the solid model of the propeller through a loft feature. Fig.4 shows the rendered output of the designed model. Rendering was performed in a commercial rendering software Luxion Keyshot 4.2 (version).

2.2 Numerical Simulation

Simulation investigation was performed satisfactorily in SolidWorks Flow Simulation by setting required boundary conditions, defining the computational domain and rotating region.

2.2.1 Model Geometry

The numerical simulation was performed using a small two-bladed propeller having a diameter of 10 in and pitch of 4.7 in. Fig.4 represents the propeller geometry. Propeller model was designed using a commercial software package SolidWorks 2013.

2.2.2 Rotating Reference

After finishing modeling, the model was inserted in flow simulation by opening a new wizard from the SolidWorks flow simulation tab. For this type of design problem, a rotating reference frame is used to simulate the effect of blade turning and generating thrust.

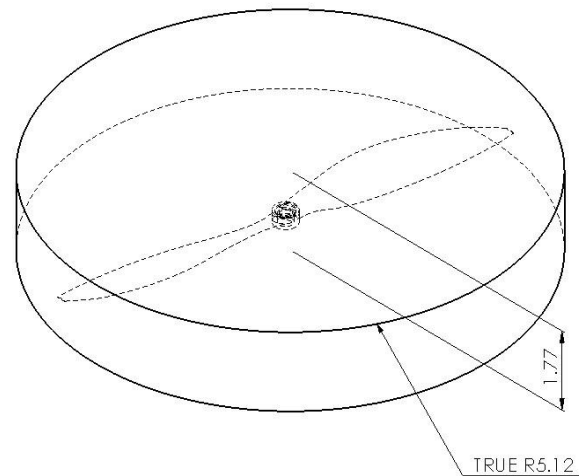


Fig.5 rotating reference frame for simulation

For this purpose a circular body should be created and the key is that the frame diameter shall be just a tiny bit larger than the blade diameter, not smaller in any case.

Axis of rotation was set about Y-axis. This illustrates that air will flow along Y-axis, when the propeller starts to rotate in order to provide a thrust force beneath. Fig.5 shows the optimum rotating frame (dimensions are in inch) used during investigation.

2.2.3 Computational Domain

The computational domain refers to a simplified form of the physical domain both in terms of geometrical representation and boundary condition imposition. For this investigation, a rectangular block surrounding the propeller acts as the computational domain.

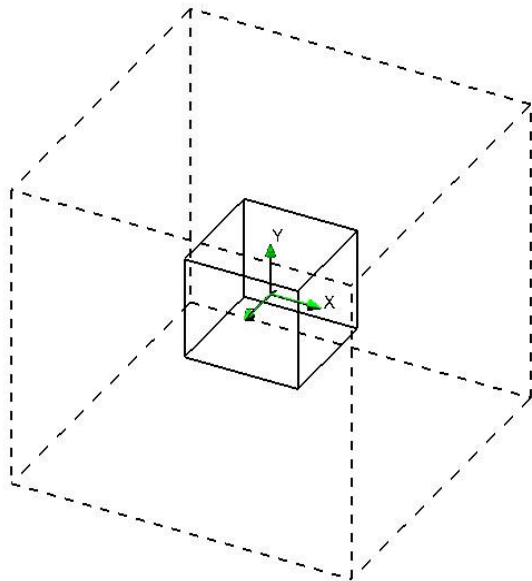


Fig.6 Simplified computational domain

Fig.6 represents the simplified computational domain, sub-domain for the propeller model in symmetrical attitude. Basically, the mathematical calculation of all fluid cells for the flow phenomena will take place in this defined domain through 3D simulation.

2.2.4 3D Mesh Generation

Effective mesh generation and update is one of the most important components of a 3D flow simulation environment capable of fast and accurate 3D computation of problems with complex geometries, including, and especially, problems with spatial computational domain changing as a function of time [8]. A 3D, solid and standard mesh generation was performed over the propeller model. Total node and element size were 15925 and 7750 respectively.

Table 1 Mesh Information

Study Name	3D Mesh
Mesh Type	Solid Mesh
Solver	Flow Simulation
Fluid Cells	2790315

Partial Cells	48527
Iterations	326
Travels	2.00126
Iterations per 1 travel	163
Refinement	326
Warnings	No Warnings
CPU time to complete mesh(hh:mm:ss)	02:06:24
Result	1 2 3 4 5 6 7 8
Resolution	



Fig.7 3D meshing over the surface of propeller blades

Proper mesh generation is an important step for simulation and Fig. 7 represents the 3D meshing of the propeller model.

2.2.5 Boundary Conditions

Necessary boundary conditions were set for the simulation purpose in the defined computational domain- at inlet, inlet free stream velocity, inlet mass flow, Mach number were imposed; same had gone for the outlet. Standard pressure and temperature were maintained for a mass flow rate of 0.0001 kg/s. Boundary layer was set to turbulent and turbulent length and intensity (I_t and L_t) were set as turbulent parameters (I-L). I_t and L_t were set 0.1% and 0.0177165 in respectively.

Initiating the boundary conditions, all the data provided will help in the mathematical calculation at each fluid cells in the computational domain. For computational algorithms applied to flow simulations, discrete boundary conditions are required. Hence, in order to receive a satisfactory output from the simulation investigation, proper boundary conditions should be defined.

3. Simulation Output and Discussion

After defining all the requirements, for different rotational speeds, thrust force was measured. The rotational speed was denoted in rpm as available in motor rating. Basically, the thrust force generated by the propeller indicates the motor power to be used in the aircraft. This phenomena ease the motor selection scenario to get a rigid flight. Thrust forces for different rotational speeds were numerically as illustrated in Table 2.

Table 2 Thrust force at different speeds

Speed in rpm	Thrust Force (N)	Maximum Value (N)	Minimum Value (N)	Delta
750	.055	.055	.044	.004
1500	.222	.224	.211	.013
2250	.508	.508	.417	.031
3500	1.221	1.221	.990	.086
4500	1.531	1.518	1.476	.109

Using the propeller data from UIUC data site [6] for the selected model 10*4.7, a graph can be plotted as shown in Fig.8 that represents the relationship among propeller radius, pitch and chord length.

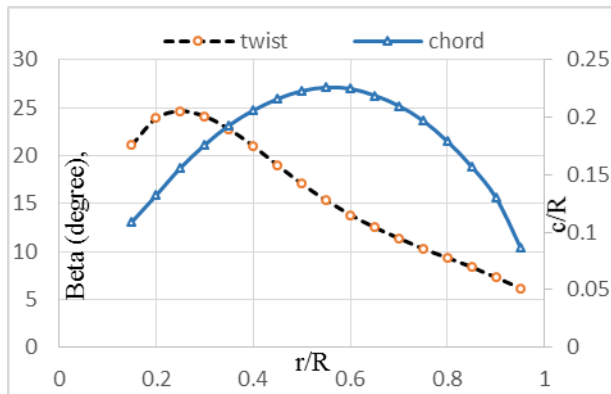


Fig.9 Graphical plot of APC slow flyer 10*4.7

Again another graph (Fig.9) using the data obtained by simulation can be plotted; thrust force (T) vs rotational speed (P).

From the graph it can be seen that, thrust force gradually increases with the rotational speed. But this increment will continue to a certain limit as propeller material has certain elastic limit. Mathematical formulation for determining the maximum rotational speed that a propeller can withstand is,

$$P = K_v * (V_{in} - V_{loss}) \quad (1)$$

Hence, maximum thrust will be obtained at maximum rotational speed.

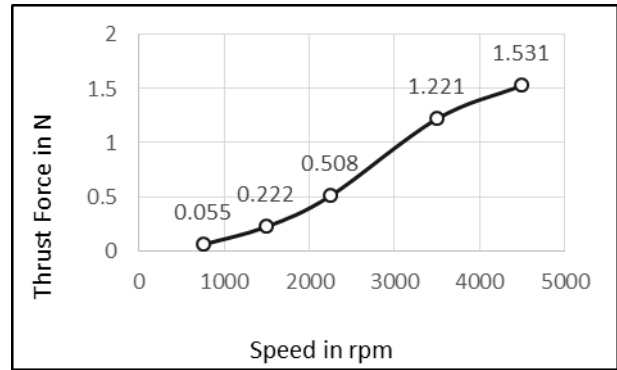


Fig.9 Thrust force at different rotational speeds

Pressure, Velocity and temperature distribution along with the velocity flow trajectory, velocity vector and velocity streamline have been represented for different rotation speeds (sample: 3500 and 4500 rpm) in the following figures by color contour. Fig.10 to Fig.14 represents the contours for speed 3500 rpm.

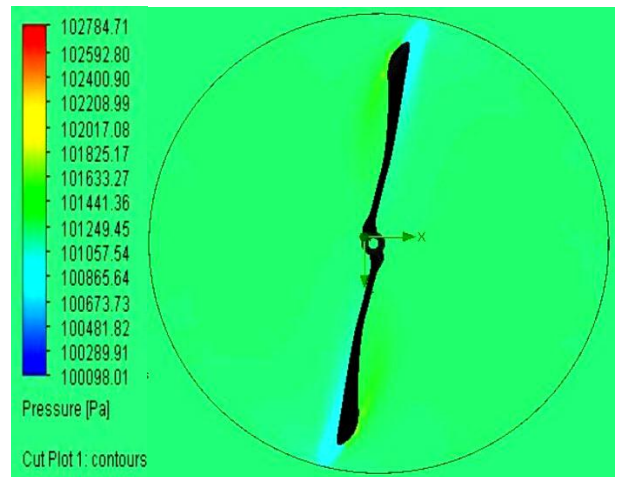


Fig.10 Pressure distribution contour

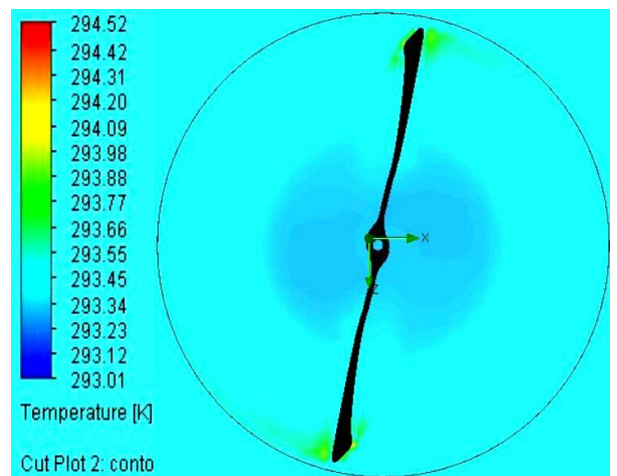


Fig.11 Temperature distribution contour

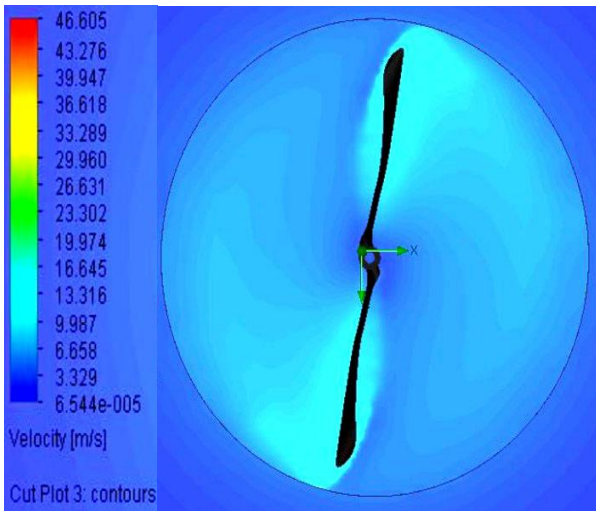


Fig.12 Velocity distribution contour

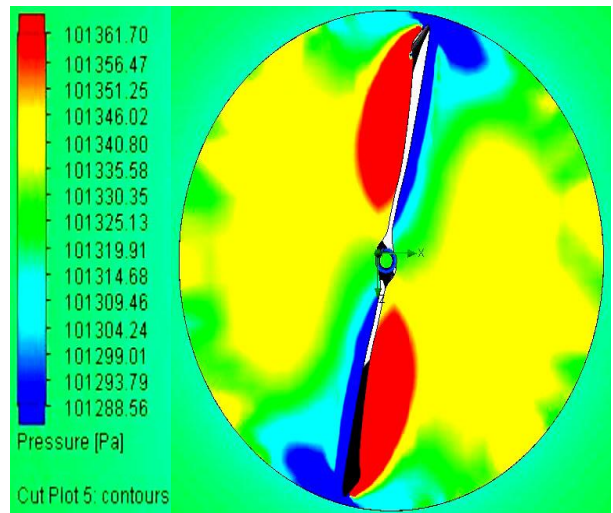


Fig.15 Pressure distribution contour

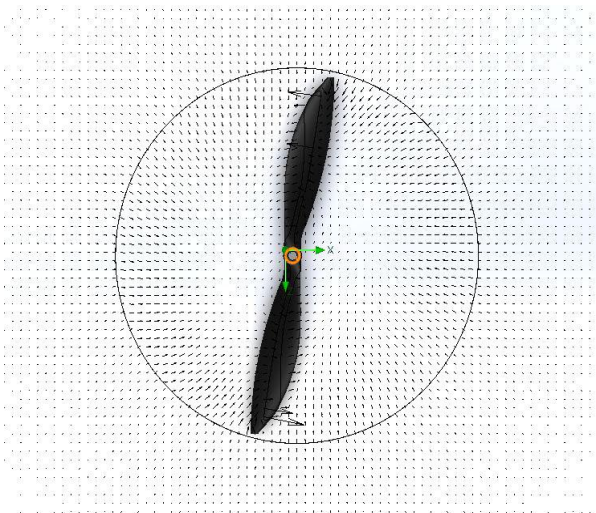


Fig.13 Velocity vector

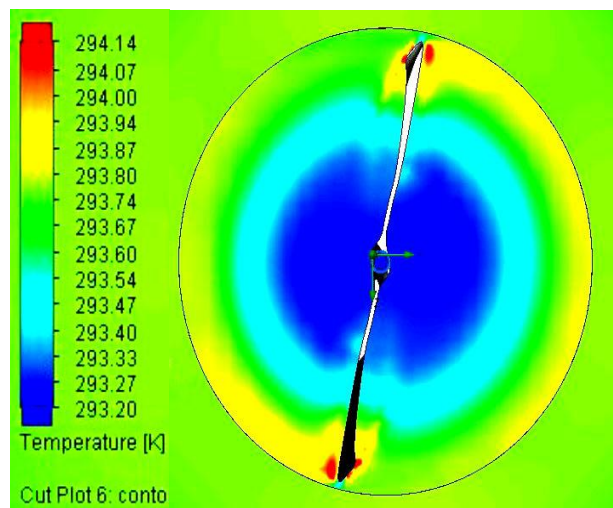


Fig.16 Temperature distribution contour

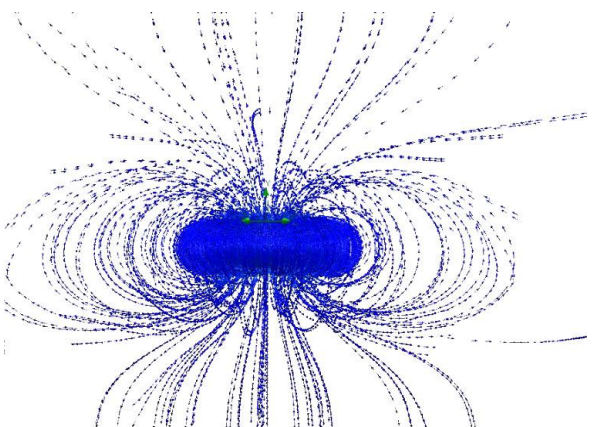


Fig.14 Velocity flow trajectory

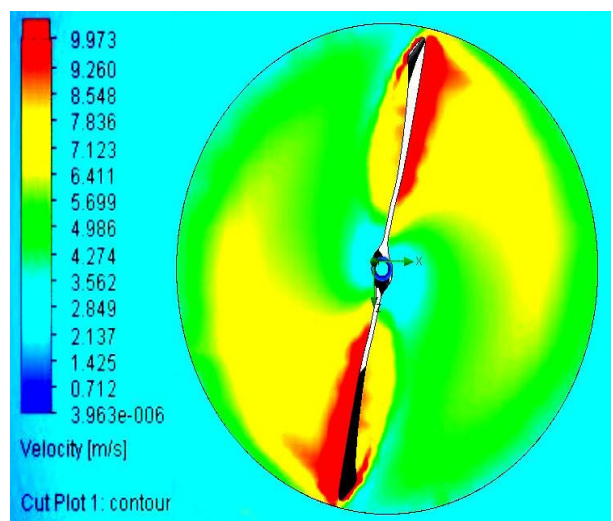


Fig.17 Velocity distribution contour

For higher rotational speeds, it was observed that, the propeller reached to its critical situation as higher torque and thrust were generated. Fig.15 to Fig.19 represents the contours for speed 4500 rpm.

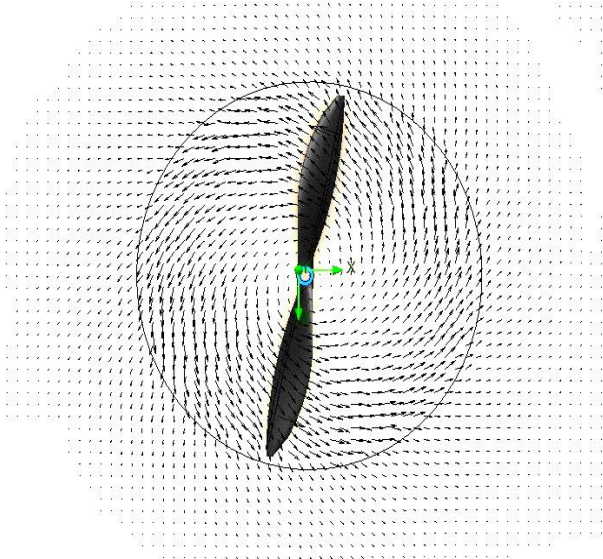


Fig.18 Velocity vector

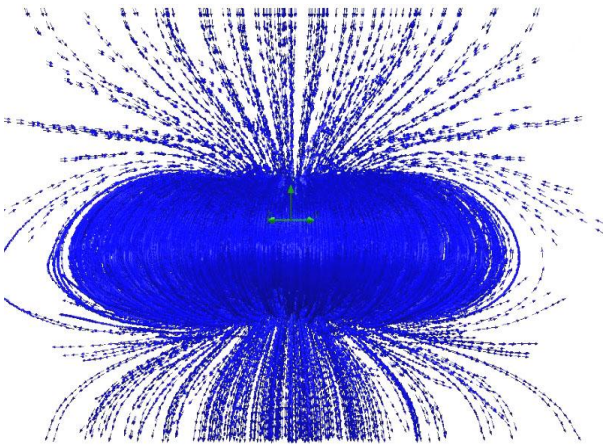


Fig.19 Velocity flow trajectory

Velocity flow trajectory shows that, due to the rotational motion, fluid (air) flows to the downward direction exerting an effective thrust force.

Simulation outputs exhibits the characteristics of the designed propeller model. It was observed that, with increasing the rotation speed, the propeller performance was satisfactorily enhanced till the critical condition. Hence, it is recommended to use the propeller in its elastic limit.

4. Conclusion and Recommendation

Propellers are considered as the main source of propulsion system in the field of radio controlled (RC) aerial vehicles. So, it is much more necessary to select the proper propeller model for the aircraft to get a satisfactory flight. Numerical simulation ease the way to expose the aerodynamics of the propeller. Here comes the main objective of this simulation investigation.

In this investigation at preliminary stage, the design fundamentals of a propeller were studied. Then, the selected model was prepared for numerical study through SolidWorks flow simulation by providing the

perquisite conditions. Hence, required thrust force at different rotational speed was determined along with other propeller aerodynamics.

Design can be modified for the optimization of burrs and vibration. Variation in computational domain, grid refinement and change in boundary conditions should be considered for the improvement in mesh generation technique.

NOMENCLATURE

- P : rotational speed, rpm
- T : Thrust force, N
- K_v : motor rating(power), kilo volt(Kv)
- V_{in} : input voltage, volt

REFERENCES

- [1] W. Shawn Westmoreland, Robert W. Tramel, and Jennie Barber, "Modeling Propeller Flow-Fields Using CFD", *46th AIAA Aerospace Sciences Meeting and Exhibit*, AIAA 2008-402, 7-10 January 2008, Reno, Nevada.
- [2] M. Abdel Maksoud, F. Menter and H. Wuttke, "Viscous flow simulations for conventional and high-skew marine propellers", *Ship Technology Research*, 45:67-71, 1998.
- [3] K. -J. Oh and S. -H. Kang, "Numerical calculation of the viscous flow around a propeller shaft configuration", *International Journal of Numerical Methods in Fluids*, 21(1):1-13, 1995.
- [4] A. Sánchez-Caja, P. Rautanhiemo and T. Siikonen, "Simulation of Incompressible Viscous Flow Around a Ducted Propeller Using a RANS Equation Solver", *23rd Symposium on Naval Hydrodynamics*, Val de Reuil, France, 527-539, 2000.
- [5] M. Stainer, "The Application of 'RANS' Code to Investigate Propeller Scale Effects", *22nd Symposium on Naval Hydrodynamics*, 222-238, 1999.
- [6] Brandt, J.B. and Selig, M.S., "Propeller Performance Data at Low Reynolds Numbers", *49th AIAA Aerospace Sciences Meeting*, AIAA Paper 2011-1255, Orlando, FL, January 2011.
- [7] Ian P. Tracy, "Propeller Design and Analysis for a Small, Autonomous UAV", *Department of Mechanical Engineering, Massachusetts Institute of Technology*, 5-6, June 2011.
- [8] A. A. Johnson and T. E. Tezduyar, "Advanced mesh generation and update methods for 3D flow simulation", *Computational Mechanics* 23(1999) 130-143, © Springer-Verlag 1999.