Numerical Analysis of Helicopter Rotor at 400 Rpm

K.M. Pandey, A. Surana and D. Deka

Abstract*—In this paper the main objective of this simulation is to analyze the flow around an isolated main helicopter rotor at a particular main rotor speed of 400 rpm, and angle of attack of 8 degrees and blades of the helicopter Eurocopter AS350B3 which uses the blade profile of standard ONERA OA209 airfoil during hovering flight conditions. For CFD analysis, the Motion Reference Frame (MRF) method with standard viscous k-ε turbulent flow model was used on modeling the rotating rotor operating in hovering flight. The Ansys fluent was used for the purpose of analysis.

***Index Terms*— Aerodynamics, CFD, helicopter, hovering, MRF, rpm.**

I. INTRODUCTION

The aerodynamics of the helicopter rotor is one of the most interesting and challenging problems faced by the aerodynamicists. The wake which is produced due to high dynamic pressure at the tip of rotor blades consists of strong vortices form and trail from each blade tip. Understanding the detailed prediction of rotor loads, performance, vibration and acoustics which also interacts with the fuselage, empennage and the tail rotor of the helicopter is critically important to design a rotorcraft. The position and the strength of the wake are influenced by many factors including blade geometry, number of blades, rotor thrust, and angle of attack of the tip path plane and operating state of the helicopter. There are three possible approaches for the accurate prediction of the wake in the helicopter: Experimentation (sometimes including real flight testing), Theoretical analysis and Computational or Simulation methods. Experimentations are currently difficult and expensive since it is very hard to measure or visualize flow on spinning blade of a rotor. Theoretical analysis has its limitations because the set of equations that govern fluid flow are so complex that they can only be solved for very simple cases. The only viable alternative is computer simulation. With rapidly increasing computer power and memory now available, it has become feasible to perform full simulations of the air flow around the rotor blade that allow engineer to accurately predict the position and strength of the rotor wake, in turn, accurately predict the performance and aero acoustics of the entire helicopter. A relative new tool in the exploration of this regime is Computational Fluid Dynamics (CFD). CFD is a technique of producing numerical solutions to a system of partial differential equations which describes the fluid flow. CFD simulations are done by discrete methods and purpose is to better understand a quantitatively and qualitatively physical flow phenomenon which is then often used to improve engineering design.

Nowadays CFD analysis is so versatile, it was used to investigate the flow about almost all types of vehicle, including rotorcraft. Long before computers were not available to perform a large number of lengthy calculations in a short period of time, aeronautical engineers used three primary methods of investigation to visualize the flow-field around a flight vehicle. The earliest methods for modeling rotors were based on an extension of Prandtl's lifting line theory for wings. In these techniques, the individual blades were modeled as line vortices, and the wake was modeled as a deformed helix. Flight tests are extremely expensive and time consuming while the solutions found are often alter rather than optimized configurations. The wind tunnel methodology can be more efficient for conventional problems such as fuselage drag reduction but many low speed interactional conditions have been found difficult to test with sufficient confidence. The helicopter industry is therefore increasingly using CFD methods by incorporating them in its design environment in order to reduce the number of wind tunnel tests and to increase the number of configurations being explored numerically.

Numerous CFD techniques were introduced for calculating the air-loads acting on helicopter blade and simulate flow over the helicopter rotors. During 1970s and 1980's, these methods were augmented by modern CFD techniques. Caradonna and Isom [1] applied the transonic small disturbance theory to lifting rotors and Chang [2] modified the full potential flow solver FLO22 for isolated wings to model rotors. The WIND code developed by NASA Glenn Research Center was used on simulating the turbulent flow past the robin helicopter with four-bladed rotor [3]. In this work, the entire configuration of helicopter was modeled with Chimera multi-block mesh and the individual blade was modeled with Chimera moving grid in quasi-steady flow-field. The k-ω SST turbulence model was used. Using the Navier-Stokes overset grid methodologies employed here, Strawn [4] showed good performance prediction for a 4-bladed UH-60 rotor. Furthermore, in this Franco-German program on simulating the isolated rotor in forward flight, ONERA Euler method uses a deforming grid strategy and DLR is based on Chimera grid method [5]. Another approach that was applied on simulating aerodynamic characteristics of helicopter rotor and that is presented in this paper is based on the simulation performed by Fluent on Eurocoter AS350B3 helicopters during hovering flight. The main rotor blade was modeled by using the Moving Reference

---

Manuscript sent Feb 18, 2012.

Dr. K M Pandey, is currently working as Professor, Department of Mechanical Engineering, NIT Silchar, Assam, India. Mobile no. +919435173130, Mail id: kmpandey2001@yahoo.com.

A. Surana, is 8th semester, B.Tech , Mechanical Engineering student at National Institute of Technology Silchar, Silchar, India, +91-9707012878, (e-mail: suranaanand@ymail.com).

D. Deka, is 8th semester, B.Tech Mechanical Engineering student at National Institute of Technology Silchar, Silchar, India, +91-9508212233, (e-mail: dhruba.yes.1@gmail.com).

Frame (MRF) method [6]. Using the standard k-ε turbulent flow model, the rotor will be simulated using MRF method and the capability of FLUENT software on simulating helicopter rotor blade in hovering flight will be evaluated.

II. LITERATURE REVIEW

Dario Fusato and Roberto Celi [7] worked on “design sensitivity analysis for helicopter flight dynamic and aeromechanic stability” which focuses on describing an efficient technique to calculate the sensitivities of bandwidth and phase delay (defined according to the ADS-33 specification), and poles of a helicopter with respect to the blade torsion stiffness GJ. The technique is based on the derivation of expressions for the sensitivities using chain rule differentiation of appropriate portions of the equations of motion. Two configurations were studied, similar to the BO-105 and the UH-60.The study showed that the semi-analytical sensitivities are in excellent agreement with the corresponding finite difference-based sensitivities, and that they are far less sensitive to step size. F or the Bo-105 configuration, phase bandwidth, gain bandwidth, phase delay, and poles, are only weakly nonlinear functions of the torsion stiffness GJ, and therefore linear approximations are accurate for broad variations of GJ. The same is mostly true for the UH-60 configuration, except that move limits are recommended for linear approximations to the phase delay at low GJ. The new technique is computationally very efficient.

González, R. Mahtani et al. [8] worked on “control and stability analysis of an autonomous helicopter” their work presented some results from the research on autonomous helicopter control conducted in the framework of the COMETS project. The paper presented both linear and non-linear control laws. A two-time scale decomposition of the helicopter dynamic has been used to analyze the dynamic behavior of the system. The fast subsystem copes with the rotational dynamics, while the slow subsystem represents the translational dynamics. The stability of the fast dynamics is demonstrated by means of a Lyapunov function. Furthermore, a feedback linearization technique was proposed to stabilize the slow dynamics. Moreover, the drawbacks of the linear control laws were pointed out and a new nonlinear control law was proposed. This control law was able to control the helicopter when large variations occur in the orientation angles and position of the helicopter.

Seawook lee et al. [9] worked on “aerodynamic analysis of the helicopter Rotor using the time-domain panel method” which focuses on unsteady aerodynamic analysis of helicopter rotor with a panel method based on potential flow theory. The panel method uses the piecewise constant source and doublet singularities as a solution. This potential based panel method is founded on the Dirichlet boundary condition and coupled with the time-stepping method. The present method was used in the time-stepping loop to simulate the unsteady motion of the helicopter rotor. And the free wake model was used for the wake simulation. The present method can be solved the three dimensional flow over the complex rotors with less computing time than commercial CFD tools. The results were well matched with the experimental results as given in the paper.

Nik Ahmad Ridhwan et al. [10] worked on “Numerical Analysis of an Isolated Main Helicopter Rotor in Hovering and Forward Flight” which mainly presented a focus on Aerodynamic characteristics of a 5-seater helicopter with different rotor configuration (i.e.; blade number and size) operating in forward flight mode which were simulated by using commercial CFD software FLUENT. The main objective of this simulation was to calculate the aerodynamic load generated by rotor during hovering and different forward flight speed range. The effects of using different rotor configuration and shaft rotational speed (different engine selection) were included in this simulation. For CFD analysis, the MRF method with standard viscous k-ε turbulent flow model was used on modeling the rotating rotor operating both in hovering and forward flight. To simulate the helicopter operating in the trim condition, the main rotor collective pitch, coning and flapping angle was calculated based on the Blade Element Theory (BET). The blade was assumed stationary and blade collective pitch, coning and flapping angle was positioned as calculated using BET. For this purpose CFD simulation has been compared with the corresponding results obtained from BET analysis and that found they were in good agreements.

Yihua Cao, Ziwen Yu [11] worked on “Numerical simulation of turbulent flow around helicopter ducted tail rotor” describing a new numerical simulation technique for the ducted tail rotor. In this paper, the flow field and performance of a helicopter ducted tail rotor in hover and sideward flight were analyzed using CFD technique. The general governing equations of turbulent flow around ducted tail rotors were first set up, and then directly solved by finite volume approach and simple approach in an axisymmetric coordinate system. The spinning fan was represented as time-averaged momentum source terms distributed along the span of the fan with functional relationship to the local flow conditions in N–S equations. The calculation method also included a stair-step representation of the shroud, staggered grid system, k–ε turbulence model and wall function method on the shroud. This unique CFD numerical simulation technique developed here not only can predict the flow field but also analyze the performance of ducted tail rotors.

According to the thesis entitled “Development and Validation of a Numerical Blade Element Helicopter Model in Support of Maritime Operations" by W.R.M. Van Hoydonck [12] which is a part of a joint research project between Romania and the Netherlands, this thesis project started with a literature survey on the simulation and handling qualities of helicopters operating near ship decks. Topics covered in this survey include the accuracy of the main rotor inflow model, dynamic ground effect, turbulence modeling, ship motion modeling and models of mathematical helicopter pilots. It was concluded that in order to increase the fidelity of the model and allow for future extensions, a numerical approach should be taken to model the main rotor blade aerodynamics. In addition, the non uniform Pitt-Peters dynamic inflow model is used for the inflow through the rotor disk. It is derived in the wind axis system, so it does not allow for sideward flight. Since one of the most often used landing procedures for helicopters stationed aboard ships includes a lateral repositioning ending in station keeping above the flight deck, the basic Pitt-Peters inflow model was modified to make this possible. The blade flapping dynamics are second order in lateral tilt, longitudinal tilt and coning, this means that the helicopter model has a total of nine structural degrees of freedom.

K.M. Pandey, G.Kumar, D.Das, D. Deka, A. Surana and H.J. Das[17] worked on , CFD analysis of an isolated main helicopter rotor for a hovering Flight, and their findings are the following. The study is done for 800 RPM.
CFD simulations are done by discrete methods and purpose is to better understand a quantitatively and qualitatively physical flow phenomenon which is then often used to improve engineering design. The main objective of this simulation is to analyze the flow around an isolated main helicopter rotor at a particular main rotor speed of 800 rpm, and angle of attack of 8 degrees and blades of the helicopter Euro copter AS350B3 which uses the blade profile of standard ONERA OA209 airfoil during hovering flight conditions. For CFD analysis, the Motion Reference Frame (MRF) method with standard viscous k-ε turbulent flow model was used on modeling the rotating rotor operating in hovering flight.

III. HOVERING ROTOR AERODYNAMICS

The rotor of a helicopter provides three basic functions: The generations of a vertical lifting force (trust) in opposition to the aircraft weight, the generation of a horizontal propulsive force for forward flight, a means of generating forces and moments to control the altitude and position of the helicopter.

The designer's considerable knowledge of aerodynamics environment in which the rotor operates and how the aerodynamics loads affect the blade dynamic response and overall rotor behavior is pretty imperative. Designers are more concerned with performance loads, vibration level, external and internal noise, stability and control, and handling qualities. Proper design of the rotor is critical to meeting the performance specification for the helicopter as a whole. Any small improvement in rotor efficiencies can potentially result in significant increase in aircraft payload capability, maneuver margins or forward flight speed. The high dynamic pressure found at the tips of a helicopter blade produces a concentration of aerodynamic forces there as a consequence, strong vortices form and trail from each blade tip. The vortices are convicted downward below the rotor and form a series of interlocking, almost helical trajectories. For the most part, the net flow velocity at the plane of rotor and in the rotor wake itself is comprised of the velocities induced by these tip vortices. For this reason, predicting the strengths and locations of the tip of the vortices play an important role in determining rotor performance and in designing the rotor. Helicopters also have a *tail rotor*, a smaller propeller on the back end of the body, generally powered by a geared drive-shaft connected to the main engine. These rotors counteract all of the undesirable torque created when spinning the main rotor by spinning in the opposite direction. By adjusting the speed of the tail rotor, you can also increase or decrease the net torque on the helicopter, causing it to *yaw*, or spin on the Y axis. This allows the pilot to easily rotate the helicopter to face the desired direction while spinning in place, without having to bank, or make a wide turn like an airplane. The combination of all of these systems allows a helicopter a full 6 degrees of freedom. It can fly forward, backward, left, right, up, down, and rotate along the X, Y, and Z axes, allowing much more control and maneuverability than any other flying machine in existence. However, if one of these systems fails, the results can be catastrophic. [13]

The rotor wake consists of a shed vortex sheet and a concentrated vortex at the tip. There is a bound circulation on the rotor blade associated with lift, and conversation of vorticity requires that the circulation be trailed into the wake at blade tip and root. The strong tip vortices are dominant feature in the rotor wake. Vorticity is also shed and trailed into the wake, creating the vortex sheet, as a result of changes in the circulation on the blade as shown in Fig. 1.

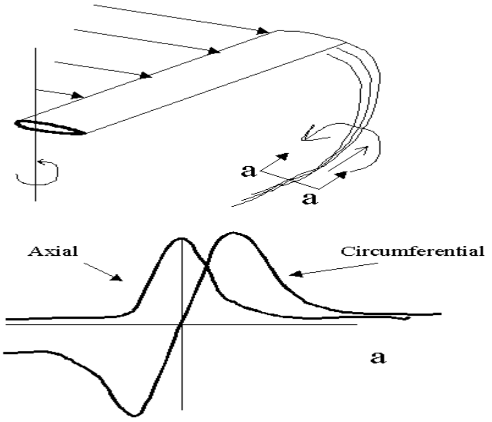

Fig. 1 Rotor Wake Formation [14]

The trailed vorticity is oriented parallel to the local free stream when it leaves the blade, similar to the tip vortex. Because of the rotation of the blade, lift and circulation are highest near the tip. Both reach a maximum before decreasing rapidly to zero at the tip, which creates a trailing vorticity of high strength at the edge of a wake to roll-up quickly into a concentrated tip vortex. Rotating blades encounter tip vortices shed from proceeding blades, which result in a phenomenon known as Blade Vortex Interaction (BVI). As the blades on the helicopter rotate, they disturb the air that pass through, causing a wake that spirals off the trailing edge of each blade. This is the wake (particularly the vortex from the blade tip) being hit by the following blade that produces the loud vibrating sound. In order to minimize the sound caused by these collisions, an accurate understanding of the BVI is essential. These unsteady loads are also an important factor in the vibration, noise and performance of the helicopter [14].

The preliminary design of the main rotor must encompass the following key aerodynamics considerations: General sizing: this will include a determination of disk loading and rotor tip speed to decide for rotor diameter, Blade platform: this will include chord, solidity, number of blades, and blade twist, Airfoil section of the blade.

The following play an important role in meeting overall performance requirement in hovering:

1. *Collective pitch angle:* Collective pitch angle changes angle of attack of all blades by an equal amount on unison .The collective pitch controls the average blade pitch. This in turn, changes the blade lift and the average rotor trust, turning the craft on its vertical axis. As drag on the blade increases with the pitch angle, this requires extra power to compensate the change in drag. The collective pitch angle is, therefore, the preliminary manifold pressure control.

2. *Tip Mach number*: A high rotor tip speed gives the rotor a high level of stored rotational kinetic energy for a given radius and reduces design weight. However, there are two important factors that work against the use of high tip speed: compressibility effects and noise. Compressibility effects manifest as increased rotor power requirements. Rotor noise also increases rapidly with increasing tip match number. Experimental work confirms that operation at lower tip Mach number is desirable to maximize hovering performance. At higher tip mach numbers performance degrades because to the increasing compressibility losses.

3. *Blade wake*: The wake from the rotating blade comprises, in part, a vortical shear layer or vortex sheet, with a concentrated vortex formed at the blade tip. The vortex sheet comprised of vorticity with vectors aligned mainly normal to and parallel to the trailing edge of the blade. Experiments have shown that blade tip vortices are almost fully rolled up within only a few degrees of blade rotation.

IV. MATHEMATICAL MODELLING

In the design environment of GAMBIT 2.3.16, the 3D design of computational domain was being designed following the steps below: The ONERA standard airfoil OA209 has been selected for the design of the rotor blades [11]. The angle of attack was fixed at 8^{o} based on a moderate normal collective pitch for a hovering flight for initial simulation design. Table I shows the rotor design configurations. Fig. 2 shows the rotor 3D view.

TABLE I: ROTOR DESIGN CONFIGURATION

SL. NO.	ROTOR ELEMENTS	CONFIGURATION
1.	Radius of the rotor disc	5.345 m
2.	Chord length	0.35 m
3.	Airfoil	ONERA OA209
4.	Number of blades	3
5.	Rotor hub diameter	1.27 m

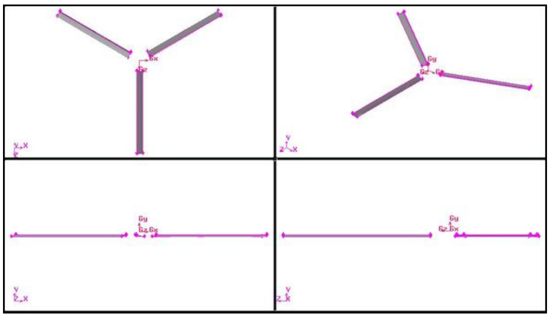

Fig 2 Views Of Rotor Section

In CFD analysis, the computational domain (or control volume) used is based on the closed-test section wind tunnel. The appropriate computational domain and selection of boundary conditions were made based on the simulation performed at Georgia Institute of Technology (GIT) on simulating rotor wake and body interaction [9]. The bigger ratio between computational domain and rotor size was used to minimize the blockage effect or wall boundary effect particularly below the rotating rotor where the airflow induced downstream by rotor. The appropriate height between rotor and the bottom wall boundary is important because it may increase the effect of ground to the rotor performance. For that reason, the helicopter rotor of radius R = 5.345 m was simulated in the computational domain the dimensions of the test wind tunnel as shown in Table II are measured from the origin of the rotor.

Table Ii: Wind Tunnel Specifications

SL.NO.	NAME OF THE PARTS OF THE DOMAIN	DIMENSIONS (m)*
1.	Upper Wall (A)	5R
2.	Bottom Wall (B)	10R
3.	Pressure Outlet Boundary (C)	20R
4.	Velocity Inlet Boundary (D)	15R
5.	Starboard Wall Boundary (E)	10R
6.	Port Wall Boundary (E)	10R

*where R is the radius of the rotor.

In pre-processing stage, the rotor was designed. The blade was meshed using edge and face meshing in order to take care of the skewness, the figure of the rotor being shown in the Fig. 3. The domain was meshed using tetrahedral meshing technique. The 3D domain is being shown in the Fig. 4.

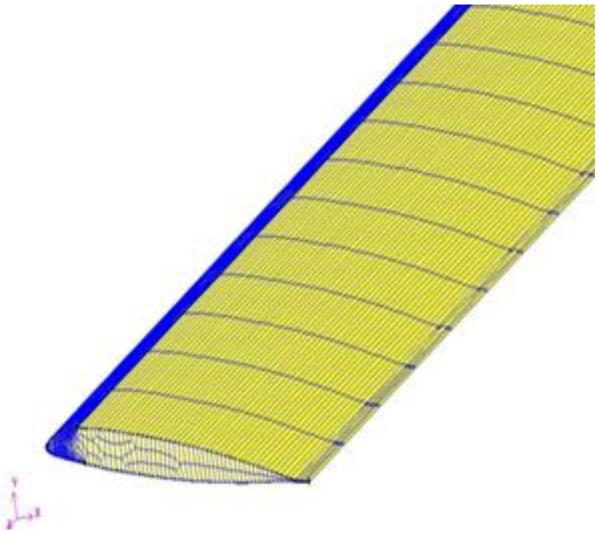

Fig.3 Meshed view of the airfoil section.

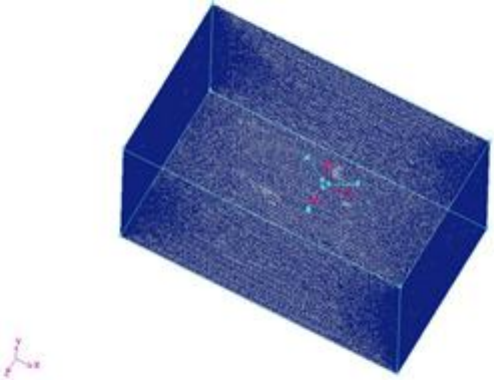

Fig. 4 Meshed view of the Computational Domain

A. The Boundary Types

The GAMBIT 2.4.6 software comes with different boundary types which synchronize with the physical conditions. The FLUENT 5/6 boundary model has been incorporated which uses the 'VELOCITY INLET' boundary types for air inlet. For the outlet conditions, 'PRESSURE OUTLET' boundary condition is used. Pressure and velocity conditions are as in the normal atmospheric conditions. The rest of the regions are defined as 'WALL' boundary type, like airfoil and domain walls.

B. The Simulation Approach

After the meshing has been completed and suitable boundary types defined in GAMBIT the model is then imported into FLUENT 6.3 using 3-D SOLVER. The finite volume approach which is used in CFD is used to create the solver. The governing equations are then integrated over the whole control volume (the computational domain being divided into finite quantity of small cells, each of which act as an individual control volume). The turbulent model used is the k-ε turbulence model suggested by Launder and Spalding. The MRF method is being used for the simulation of fluid flow over the rotor blades during hovering flight condition.

C. Input Considerations

The simulation was carried out in the wind tunnel under normal atmospheric conditions using MRF methodology for varying RPMs of the rotor blades. The input parameters used in the simulation study are presented in Table III.

TABLE III: PARAMETERS USED IN SIMULATION

SL. NO.	PARAMETERS	VALUE
1.	Operating condition	101325@298K
2.	Velocity inlet	3 m/s
3.	RPM	400

V. RESULTS AND DISCUSSSIONS

The flight characteristics obtained for 400 RPM was studied by using the CFD package FLUENT 6.3 and the behavior of the parameters like velocity and vorticity were obtained after simulation.

A. Contours of Velocity Magnitude

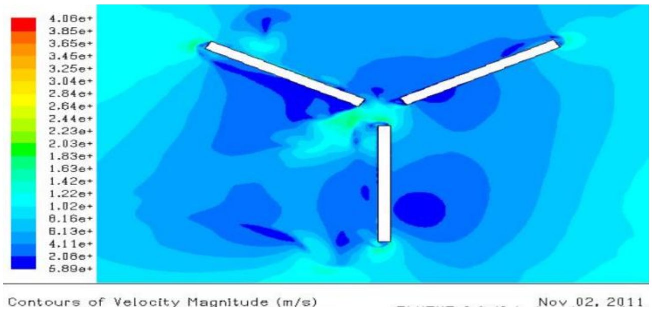

Fig 5 Velocity Magnitude Contour at 400 RPM

Fig. 5 shows that velocity field is gradual, wake formation is not too high and tip vorticity is minimum for 400 RPM main rotor speed.

B. Contours of Vorticity

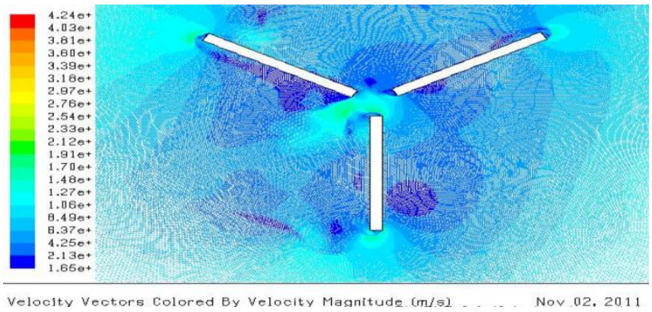

Fig. 6 Vorticity by Colours of velocity Magnitude at 400 RPM

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

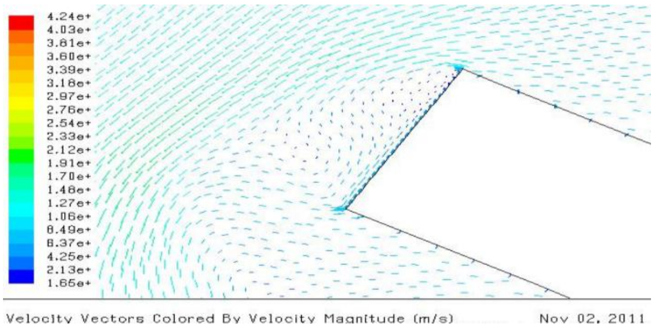

Fig. 7 Vorticity (close view) by Colours of velocity Magnitude at 400 RPM

As shown in Fig. 6 and Fig. 7, the vortex formation as shown by velocity vectors is minimum, as no exact vortex zones are formed around the rotor tip, which is in alignment with the results obtained experimentally for the helicopter Eurocopter AS350B3.

VI. CONCLUSION

From the above analysis, it can be concluded that a main rotor speed of around 400 RPM is suitable for the hovering flight conditions for the helicopter taken into account, that is Eurocopter AS350B3. The computational results for this helicopter show that the safe optimum range for this helicopter is around 380-430 RPM for hovering flight conditions. The CFD Analysis performed in FLUENT also shows that this is the optimum speed. There is a further need to verify the computational results with the experimentations.

ACKNOWLEDGEMENT

The authors acknowledge the financial help provided by AICTE from the project AICTE: 8023/RID/BOIII/NCP(21) 2007-2008 .The Project id at IIT Guwahati is ME/P/USD/4.

REFERENCES

[1] Caradonna, F. X. and Isom, M. P. (1972). "Subsonic and Transonic Potential Flow over Helicopter Rotor Blades", *AIAA Journal, No. 12, pp. 1606-1612.*

[2] Chang, I.C. (1984), "Transonic Flow Analysis for Rotors", *NASA TP 2375.*

[3] Xu, M., Mamou, M. and Khalid, M. (2002). "Numerical Investigation of Turbulent Flow Past a Four- Bladed Helicopter Rotor Using *k-ω* SST Model", *The 10th Annual Conference of CFD Society of Canada*, Windsor.

[4] Strawn, R. C. and Djomehri, M. J. (2001). "Computational Modeling of Hovering Rotor and Wake Aerodynamics," *American Helicopter Society 57th Annual Forum, Washington, DC.*

[5] Sides, J., Pahlke, K. and Costes, M. (2001). "Numerical Simulation of Flow Around Helicopter at DLR and ONERA", *Editions Scientifiques et Medicales Elsevier.*

[6] FLUENT News 2002 (11)2, pp: s9

[7] Dario Fusato, Roberto Celi (2001). "Design sensitivity analysis for helicopter flight dynamic and aeromechanic stability", 57th Annual Forum of the American Helicopter Society, Washington D.C.

[8] A. González, R. Mahtani, M. Béjar, A. Ollero "control and stability analysis of an autonomous helicopter"

[9] Seawook Lee, Hyunmin Choi, Leesang Cho, Jinsoo Cho "Aerodynamic analysis of the helicopter rotor using the time-domain panel method", ICAS 2010, 27th international congress of the aeronautical sciences.

[10] Nik Ahmad Ridhwan, Nik Mohdi and Abas Ab. Wahabii. "Numerical Analysis of an Isolated Main Helicopter Rotor in Hovering and Forward Flight."

[11] Yihua Cao, Ziwen Yu. "Numerical simulation of turbulent flow around helicopter ducted tail rotor", Aerospace Science and Technology 9 (2005) 300–306

[12] W.R.M. Van Hoydonck(2006). "Development and Validation of a Numerical Blade Element Helicopter Model in Support of Maritime Operations."

[13] A document on the internet "Unity3D Helicopter Tutorial". [http://activeden.net/item/rc-helicopter-simulation/236149]

[14] An image on the internet [www.emeraldinsight.com/journals.htm?articleid=1913570]

[15] Froude RE. Trans Inst Naval Architects,1889;30:390.

[16] Rankine WJM. On the mechanical principles of the action of propellers. Trans Inst Naval Architects (British), 1865;6(13).

[17] K.M. Pandey, G.Kumar, D.Das, D. Deka, A. Surana and 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. 2, No.1 Feb,2012, PP.131-137

AUTHOR'S PROFILE

Dr. K.M. Pandey did his PhD in Mechanical Engineering in 1994 from IIT Kanpur. He has published and presented more than 200 papers in International & National Conferences and Journals. Currently he is working as Professor of the Mechanical Engineering Department, National Institute of Technology, Silchar, Assam, India. Department, National Institute of Technology, Silchar, Assam, India. He has also served in the capacity of head from July 2007 to July 2010. He has also worked as faculty consultant in Colombo Plan Staff College, Manila, Philippines as seconded faculty from Government of India. His research interest areas are Combustion, High Speed Flows, Technical Education, Fuzzy Logic and Neural networks, Heat Transfer, Internal Combustion Engines, Human Resource Management, Gas Dynamics and Numerical Simulations in CFD area from Commercial Softwares. At present, he is serving the institute in the capacity of Dean, Faculty Welfare since November 2011.

A. Surana and **D. Deka** are currently studying in 8th semester, B.Tech inMechanical Engineering at NIT silchar, Assam, India.