

Helicopter Main Rotor aerodynamic simulation with CFD

GAPR Perera[#], HDH Jagathsinghe, SKK Dilshan, MAM Sudaraka, SLMD Rangajeeva

Department of Aeronautical Engineering, Faculty of Engineering, General Sir John Kotelawala Defence University, Sri Lanka.

[#] pramith.27@gmail.com

Abstract— The Bell 212 helicopter is one of the major general purposes utility rotorcraft currently in use in SLAF which is containing NACA 0012 aerofoil in the two blades of main rotor. The primary objective of this research project is to evaluate the aerodynamics behavior of the Bell 212 main rotor in hovering, forward and vertical flight by conducting Computational Fluid Dynamics (CFD) analysis and make comparison with the theoretical solutions in both Blade Element and Momentum theories. The CFD simulations have been done in subsonic flight regimes only. As a means of validating the results, the CFD analysis has been done with turbulence model. CFD analysis of a rotating blade was achieved by using the rotating mesh. A 1:1 SOLID WORKS solid model (14.201m) of Bell 212 main rotor was used to generate computational mesh and subsequent CFD simulations were performed mainly with ANSYS Fluent. A surface and volume mesh continuum was generated that contained approximately seven million polyhedral cells, where the Finite Volume Method (FVM) was chosen as a discretization technique. During the post processing phase of the CFD results, the aerodynamic characteristics of the Bell 212 main rotor have been predicted in terms of main rotor speed of 800rpm for hovering forward and vertical flight with angles of attack of the blades to 2°. In CFD simulation process, results were recorded such as rotor RPM, Tip velocity, Mass flow rate, Heat flux, Temperature changes and total and static pressure variation over and around the main rotor. Throughout those values comparisons have made with theoretical aerodynamic results under Blade Element theorem and Momentum Theorem. An implicit unsteady flow solver, with an ideal gas and a SST (Mentor) K-Epsilon turbulence model were used. Hover, vertical and forward cases were examined. Several aerodynamic forces such as Coefficient of Drag, Coefficient of Lift and Coefficient of Momentum were estimated during the Fluent analyzing process in CFD. The flow physics revealed with CFD analysis are well aligned with selected main rotor practical theories in both Blade Element and Momentum theories in hover, vertical and forward flight maneuvers. The forecasted values of aerodynamic parameters for Bell 212 main rotors are little bit different than expected. It has been found this particular fact is directly related to computational limitations associated with CFD. The final outcomes of this

research will provide a better guide in future CFD analysis on the helicopter main rotor analysis since it is more valuable as a rotating meshed CFD research.

Keywords—Bell212, CFD, Turbulence models

I. INTRODUCTION

A. BELL 212 HELICOPTER

Model 212 is a single pilot, fifteen place twin engine helicopter with a two-blade semi-rigid, main rotor and a two bladed tail rotor that provides directional control. It was chosen as project model to access data easily. It has two bladed, semi rigid flapping type main rotor. Each blade is connected to a common yoke by a blade grip and pitch change bearings with tension straps to carry centrifugal forces. Allowing rotor to flap, main rotor has a mast with a bearing mounted trunnion. Blade pitch change is accomplished by movement of collective and a series of controls terminating at blade grip horn. Ascending helicopter is done by upward movement of collective, which help to increase angle of attack of rotor blades. Downward movement of collective causes descending the helicopter. Movement of cyclic controls the tilting of the helicopter. Tail rotor has a two bladed system, mounted on right side of vertical fin. Helicopter main rotor is the major component of any rotorcraft which gives the main rotational power to the aircraft to generate the lift force and make it fly. Also it helps to do all the rotorcraft maneuvers such as hover, vertical and forward flight in its operation. Main rotor diameter is 14.69m (48ft). Max thickness 12% at 30% chord. Chord length of the NACA 0012 is 23.38 inches. At low Reynolds numbers, this airfoil's characteristics will be unsteady, with fluctuations in lift, drag, and pitching moment. Study the selected rotorcrafts' main rotor in both Blade Element and Momentum theorem with the actual values and do the CFD simulation for the same will be nominated as the research problem. The results from both ways will be compared at the end of the research. Future actions can be taken to improve the quality of the results of the research. Final findings of this research were helped to develop the model designing and CFD simulation knowledge.

B. CFD

Computational fluid dynamics is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. CFD is particularly dedicated to the fluid, in that motion and how the fluid flow behavior influences process, that may include heat transfer and possibly chemical reactions. The physical characteristics of the fluid motion can usually be described through fundamental mathematic equations usually in partial differential form which govern a process of interest and are often called governing equations in CFD. In order to solve these mathematical equations, they are converted into computer programs or software packages using high level computer programming languages. The CFD is based on the Navier-Stokes equations. These equations describe the pressure, temperature; velocity and density of a moving fluid are related.

C. TURBULENCE MODELS

Turbulence is a type of fluid flow pattern that is with random and chaotic three-dimensional vorticity. With the present of turbulence it results the increase in energy dissipation, mixing, heat transfer and drag. With turbulence the simple key words like skin friction and heat transfer coefficient and complex as energy spectra, turbulence fluctuation magnitudes and scales. The complexity of the mathematical models will increase with the amount information required about the flow field. Rotational, three-dimensional and time-dependent are the key elements of turbulence. Several turbulence models generally are in use in common CFD simulations. Turbulence models are classified according to the governing equations which they are applied to and also with the number of transport equations which uses to solve the models. Eventhough it is easy to solve dimensional problems through CFD analysis every problem won't get the most accurate results in real physical conditions.

Reynolds-Averaged Navier-Stokes model is the most commonly used model in production application. This is categorized according to the wall function, number of variables used and what these variables are. During this research, there were mainly focused models named,

- a. Spalart-Allmaras
- b. K-Epsilon(ϵ) Model
- c. K-Omega(ω) Model

II. METHODOLOGY

A. SOLID MODELING

The 1:1 solid model of the main rotor was created using Solid Works 2014 drawing software. The XYZ coordinate

file of NACA 0012 aerofoil was found and it was download from the Aerofoil.com website. Those coordinates were converted into the design enable format by using MS Excel 2010 and modified them as three axis coordinates with zero Z coordinates. This helped the design to model the aerofoil in XY plane in Solid Model. After importing the .dat file of the coordinates properly to the Solid Works modeling platform, the design part was started in the blade using Front Plane. By using Extrude feature in Solid Works, model was embossed with aerofoil as the 3D blade into the respective blade length. Using more than 145 coordinates helped us to model the outer surface of the aerofoil very smooth. The blade length, chord and other dimensions of the main rotor as mentioned in the Table 1 and 1:1 scaling was used. Close observation was required every time while modeling, about the every drawing of the model as they were indicated as Closed Entities. This requirement was helped later on to develop the mesh in Gambit without any delay or error.

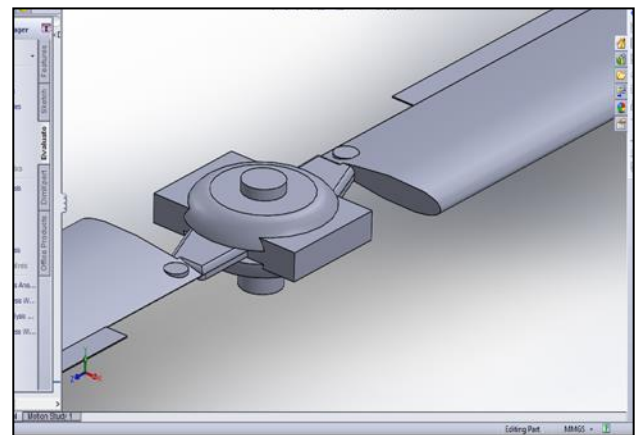


Figure 1: Solid Model

B. MESH GENERATION

The second step was importing the drawn IGS file of the model into the Gambit working surface. Import IGS file command was used to import the model into the Gambit itself. Main Rotor model which created contained more than 50 faces, more than 200 vortex points and around 145 edges. These values were analysed after importing it successfully into Gambit.

The faces, vortexes and edges containing of the model were converted as the common volume before the meshing process. This step was taken to avoid the errors in mesh type selection and reduce the meshing time. Several attempts were taken to convert this model into single volume as the process indicated some errors in command bar as "Selected Vortex/Points can't be converted to Volume". The error was found with the solid model designing phase and corrected the drawing by closing some gaps which have not connected. In 3D modeling with Gambit, it must be essential to follow the volume subtract method to develop the mesh for any

model. After converting the faces into a common volume it was renamed the model as Main Rotor and indicated further as a Volume. Time to time, few inspections has been made to check the model in Wire Mode to find whether any un-linked edges or points exist or not. The origin was automatically selected as the center of the rotor hub by the Gambit. The Coordinate System1 which was previously given in IGS exporting process of Solid Works, was selected as the same origin point by the Gambit software. It was converted to a movable model around the origin after converting it to a Volume object. The parameter values for main rotor mesh and domain mesh is given in Table 1 below.

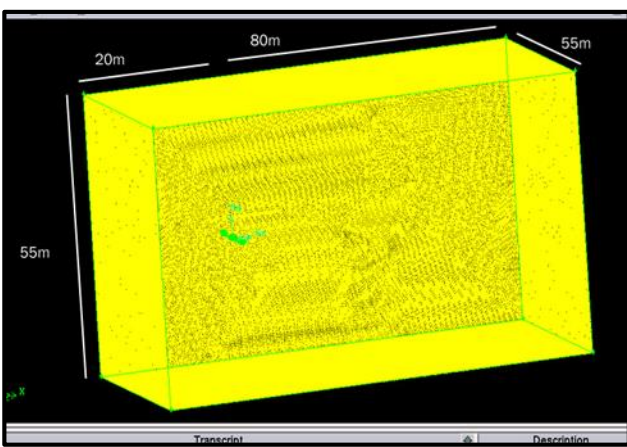


Figure 2: Generated Mesh for FWD flight

C. CFD SIMULATION

The next and the most important process of this project was running the simulation in Ansys Fluent 6.3 solver software. After exporting all the meshed volumes as 3D .msh file formats from the Gambit, they must be opened and read in Fluent. The mesh was reordered, to generate the mesh more refined. Then the models, materials and boundary conditions were set. There was an initial checking process for every solving in Fluent which was used before running the simulations. The version for simulations was selected as “3ddp” to get the simulation result in very high accuracy for 3D model, which was created.

1) TURBULENCE MODELS:

The selected turbulence model for running the simulation for hover was $k - \epsilon$ with pressure based solver. It was the most famous turbulence model ever used in CFD researches. Also it was a 2 equation based turbulence model and accuracy level was high. The flow was selected as steady state by assuming the flow properties were not changing with respect to the time. The velocity formulation was set as “Absolute” to get the results independently for each aerodynamic force.

2) MATERIALS:

The working fluid for this simulation was selected as “air” an ideal gas and assumed that the main rotor operating in the standard atmospheric conditions. It was considered the standard values as 101325 Pa and 300 K temperature.

3) BOUNDARY CONDITIONS:

Boundary conditions were set upon the selected boundary types given in the Gambit process after creating the mesh. Fluent was the software which was used to adjust and solve the governing equations. Properties and values were changed inside fluent according to the flight condition used in this research. The box domain which was used for all three simulations was subjected to the boundary conditions as reference

Table 1: Mesh features

Model	Hover	FWD flight	HIGE	HOGE
Cells	875398	876382	621032	612358

Table 2: Boundary conditions

Volume	Specify Continuum Type	Specify Boundary Type
Main Rotor (2 blades , Hub)	SOLID	WALL
Hovering		
Left Face of the Box Domain	SOLID	WALL
Right Face of the Box Domain	SOLID	WALL
Upper Face of the Box Domain	FLUID	VELOCITY INLET
Lower Face of the Box Domain	FLUID	PRESSURE OUTLET
Forward Flight		
Left Face of the Box Domain	FLUID	VELOCITY INLET
Right Face of the Box Domain	FLUID	PRESSURE OUTLET
Upper Face of the Box Domain	SOLID	WALL
Lower Face of the Box Domain	SOLID	WALL
HIGE / HOGE		
Upper Face of the Box Domain	FLUID	VELOCITY INLET
Lower Surface of the Box Domain	FLUID	PRESSURE OUTLET
Right and Left faces	SOLID	WALL

4) SOLUTION:

The solution controls for this hover simulation were set with COUPLED algorithm method which has linked the velocity and the pressure equations together. The higher accuracy and quality of the results were expected by selecting this algorithm. This method is ideal for simulations designed for aerodynamic scenarios, since it has got pressure-velocity equation solving in steady state flow. The results were initially evaluated and then

corrected by the COUPLED method. The Courant number was set to 4.

The Under –Relaxation Factors have set as follows and the Discretization set for Second Order Upwind except the Pressure, since it has got rather accuracy than the First Order Upwind. The initialization values were given for hover simulations in Fluent for selected variables. By giving the “All Zones” for compute From command, it has taken the entire volume domain for the simulation. The Solution Initialization panel was allowed to set initial values for the flow variables velocity in three x, y, z components, Temperature, Turbulent Kinetic Energy, Heat Dissipation Rate and Gauge Pressure. Then the solution was initialized using the given values. After setting the standard values for pressure and temperature it was fed to the settings by command “Init” and “Apply” respectively.

Monitoring the Residuals was set in Residual Monitors window of Fluent. The Operations was tick as Plot in order to plot the Iteration Vs Residual convergence. The Convergence for selected aerodynamic forces has shown dynamically during the simulation running. The Convergence was checked of Continuity Residual only. Absolute criteria have changed three times up to 1×10^{-7} when the simulations stopped at Convergence started to appear for respective aerodynamic forces.

5) REFERENCE VALUES:

The reference values for all three simulations were same except the velocity values and respective mass flow rates as 0 m/s for Hover and HIGE/HOGE and 35 m/s for forward flight. Working flow air density was set to 1.225 Kg/m³, Enthalpy was 113858.2 j/Kg , Temperature as 298K and the viscosity was 1.7894×10^{-5} Kg/m-s

7) ITERATE:

The Iteration window provided all the information on iterations prior to start the simulation running. The number of iterations were set as 10,000 to be performed for all three simulation processes. The Convergence has occurred at average value around 5000 in all three simulations for Drag, Lift and Momentum graphs against the iterations. Residuals were set at 1×10^{-6} .

III. RESULTS AND DISCUSSION

A. SOLUTIONS FOR HOVER

The initial simulation was done for Hover flight condition of the isolated main rotor. Velocity and the other values are as given as in early chapter. The aerodynamic forces and the tip velocity variation on the main rotor were closely observed whether those results mapped with the theoretical concepts used in helicopter aerodynamics. CFD results were recorded in three coefficients.

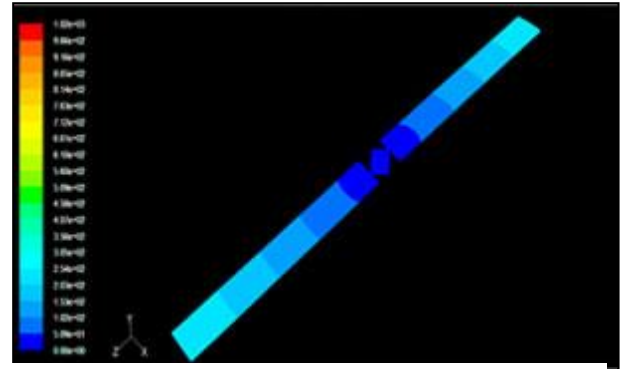


Figure 3: Velocity Variation over the blade during Hover

Pressure distribution over the blades and pressure variation around the main rotor was observed as contour displays.

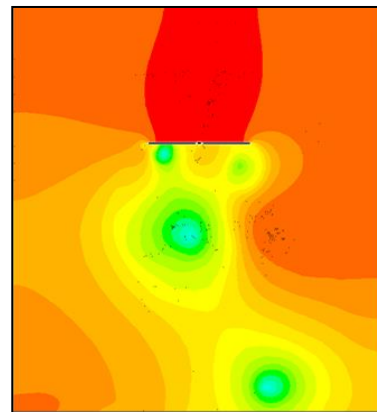


Figure 4: Pressure Distribution around the main rotor during hover

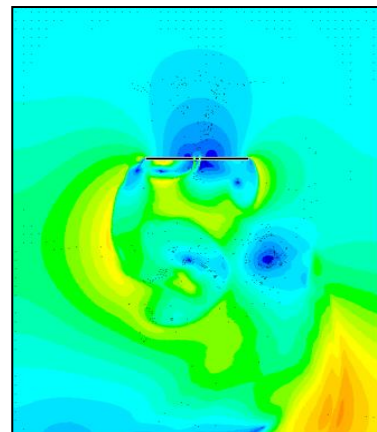


Figure 5: Velocity variation around the main rotor during hover

B. SOLUTIONS FOR FORWARD FLIGHT

After setting the flow properties and boundary conditions for forward flight conditions the simulation was run in Fluent. The contour displays of pressure and velocity were obtained. Flow properties and boundary conditions allocated for the domain were same as Hover flight except the velocity and mass flow rate. The forward flight also

happened in standard atmospheric conditions. The Flapping angle was set as 5 degrees. The pressure variation around the main rotor and the pressure distribution on the blades at Forward Flight was obtained with the simulation results as per the reference. Pressure variation was not much more prominent on the blades during forward flight.

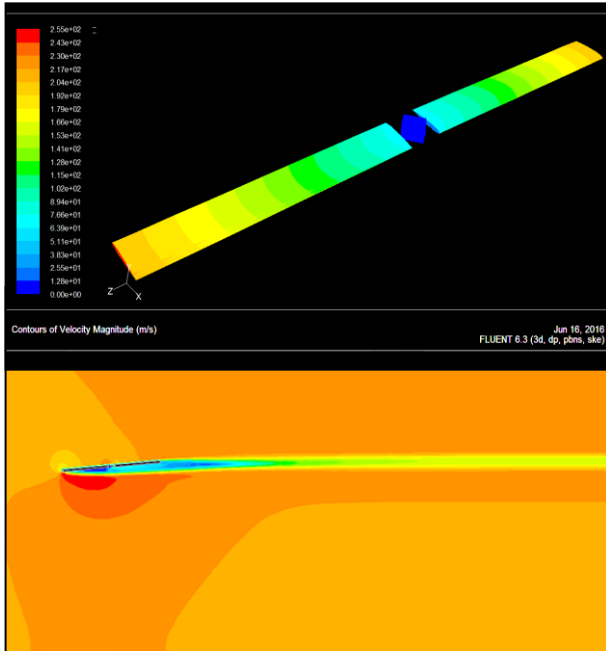


Figure 7: Velocity Variation around the main rotor during FWD flight

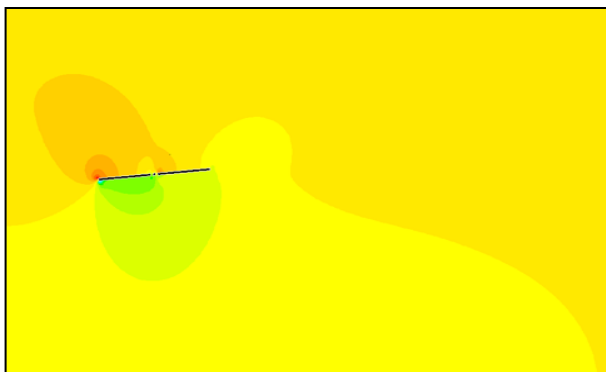


Figure 8: Pressure distribution around the main rotor during FWD flight

Table 3: Convergence values

SIMULATION	CONVERGENCE STARTED AT	TOTAL ITERATIONS RUN	ABSOLUTE CRITERIA
Hover	3874	5600	1×10^{-6}
Forward Flight	2921	5035	1×10^{-6}

HIGE	4234	5000	1×10^{-6}
HOGE	4628	5000	1×10^{-6}

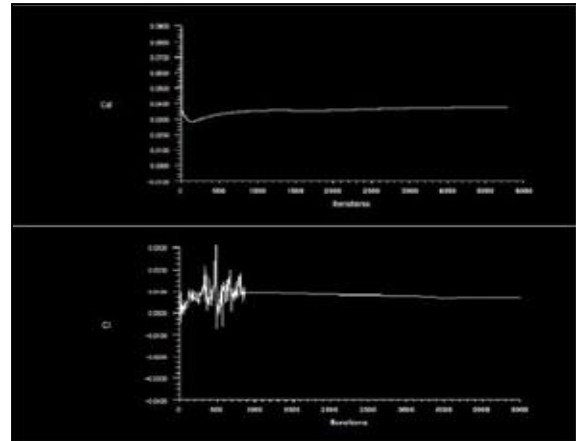


Figure 9: Convergence history for hover

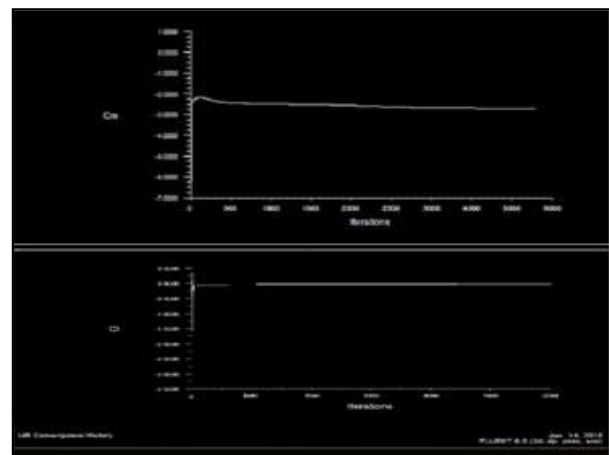


Figure 10: Convergence history for forward flight

C. SOLUTIONS FOR "HIGE" AND "HOGE"

Hover In Ground Effect (HIGE) and Hover out of Ground Effect (HOGE) are two of major flight conditions in helicopter aerodynamic as effects of those are directly affected to the changes of the aerodynamic parameters of the flow around the main rotor. As per the theoretical concept proven for HIGE and HOGE, the low density of vortices was created in HIGE and this guided to reduce the drag acted on the main rotor. This lower drag condition will be reduced the power requirement during hover in ground effect (HIGE).

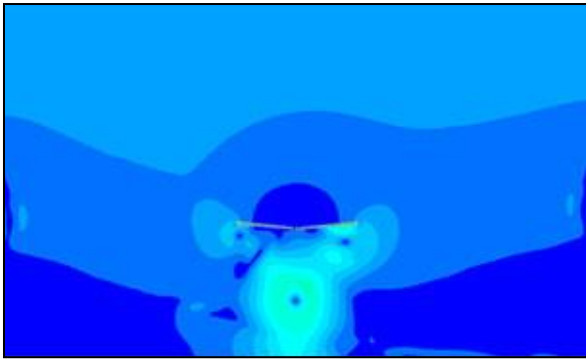


Figure 11: Velocity Variation around the main rotor during HIGE

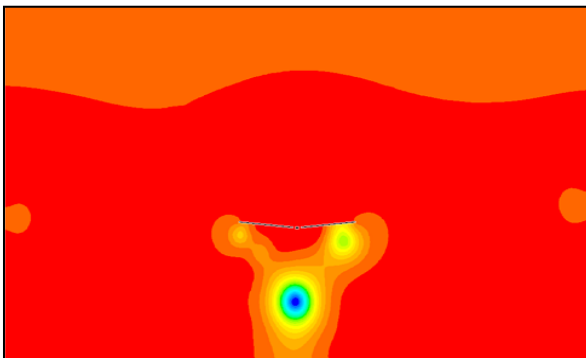


Figure 12: Static Pressure distribution around main rotor during HIGE

The Hover out of Ground Effect also generated the graphical displays in velocity and pressure variation. The vortex generation is much high and strong around the main rotor during HOGE as follows. So the main engine will need more power to run the main rotor during HOGE to overact the high drag acted on the helicopter.

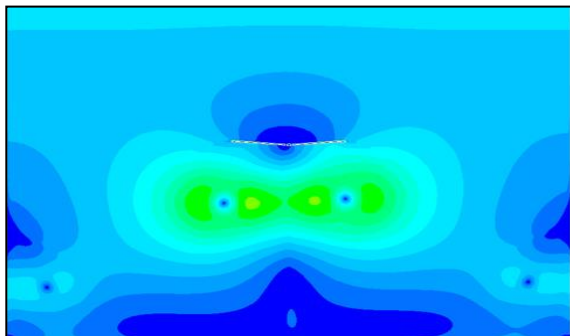


Figure 13: Velocity variation around the main rotor during

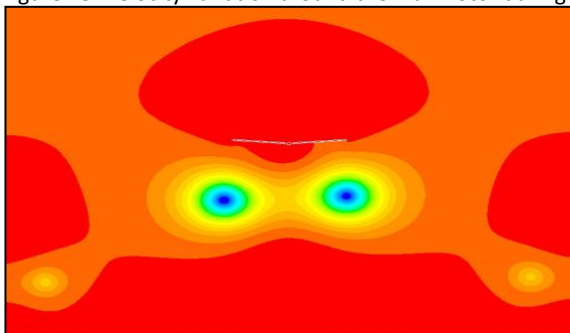


Figure 14: Static Pressure variation around the main rotor during HOGE

D. CALCULATIONS

Theoretical analysis of Bell 212 helicopter main rotor has done with the available data and parameters. Maintenance manual of Bell 212 has used to get the data and values. All calculations were done according to two theories named Blade Element Theory and the Momentum Theory. Several assumptions were taken to support both the calculations and simulations, since the main rotor was isolated in this research. Considerations and analysis were based only on the main rotor as it was taken as the isolated main rotor. Neglect tail rotor rotational effect, entire aircraft is empty and isolated main rotor were general assumptions taken for the calculations and analysis.

During CFD simulations no any weights have taken into account. This effected for power calculations of the model main rotor except the flow pattern analysis. The key assumption of this research was, all three flight maneuvers were done by the helicopter in standard atmospheric conditions. Inlet flow velocity values have taken as 0 m/s for hovering, 35 m/s for forward flight and 0 m/s for HIGE and HOGE. The Flapping Angle was set at 5 degree. The slipstream from the rotor exerts a download force on the helicopter fuselage. Thus in the hover the main rotor must generate sufficient thrust to support not only the weight, but balance this download force. Also in a vertical flight an additional thrust is required to overcome the download effect. In Momentum Theorem it cannot be explained about the drag acted on the helicopter as it was assumed the flow is inviscid. Therefore the Blade Element theory was used to describe about the drag. Download force will be effected only for Vertical and Hover flight.

Table 4: Data used for Calculations

Main Rotor Radius	48 ft (14.69 m)
Blade Chord	0.593852 m
Number of Blades	2
Down load	5% (Assumed Value)
Rotorcraft Empty Weight	2961.7kg
Fuselage Drag Area	4.2 m ² (Assumed Value)

The simulation results obtained from the CFD, can be analyzed graphically with the calculations and the theoretical concepts about the flow patterns acting around the main rotor. Several facts were missed in CFD model which were compulsory for the helicopter aerodynamics calculations such as the effects of fuselage weight for the drag and tail rotor effects for the flight. General CFD results have matched with the actual phenomenon which described in rotorcraft aerodynamics.

Forward flight condition of the Bell 212 helicopter have obtained from the CFD simulations by considering it as an isolated main rotor. Coefficient of lift, drag and momentum were checked with the converge values obtained from CFD with the theoretical calculations.

IV. CONCLUSION

This research project was conducted mainly to analyze the selected Bell 212 main rotor under two main helicopter aerodynamic theories named Blade Element theorem and the Momentum theorem. The actual results were compared and analyzed further with real scale model of the main rotor with CFD simulation for Hover, Forward flight and HIGE and HOGE flying maneuvers. The CFD results we obtained from this research were based on a 1:1 scaled solid model of the main rotor designed by using Solid Works, 3D mesh developed by using Gambit and the CFD simulations run in the Fluent.

Several deviations have been observed after comparing the results with the real data calculations for the respective helicopter. This happened due to non consideration of the fuselage weight and effects of the tail rotor in the CFD simulations. General flow patterns and pressure velocity distributions have been recorded by using the same RPM value (324RPM) for Forward flight and Hover flight. The k-ε turbulence model was used to run the simulations. Convergence of Cd, Cl and Cm values were recorded against the iterations. Separate two velocities have been used to separate flight maneuvers. Practical issues occurred during the simulation and mesh development process due to computer memory limitations in this project. Since this was a 3D CFD project, it has taken long time for pre-processing and simulations. Calculations for power and thrust for the selected rotorcraft were completed with the aid of the flying manual of the Bell 212 helicopter. The results obtained from the simulations were compared in three main aerodynamic coefficient for forward and hover flight. The property changes were observed with well alignment with the theoretical concepts about the rotor blade aerodynamics. The results from the CFD simulations were analyzed deeply with the Blade Element theorem and Momentum theorem and recommendations were suggested with proper findings and recorded for further analysis.

Table 5: Results Comparison

Parameters	Theoretical Results	CFD Results
C _L for hover	0.0094685	0.0086
C _L for FWD flight	0.0884628	0.0756
C _D for hover	0.0058	0.0382
C _D for FWD flight	0.0056	0.0021
C _M for hover	0.007	-2.53
C _M for FWD flight	0.009	0.085
V _T for hover	248.36ms ⁻¹	254ms ⁻¹
V _T for FWD flight	248.36ms ⁻¹	217ms ⁻¹

REFERENCES

R. E. Bensow, "Simulation of unsteady propeller Blade using openFoam," M ulheim, 2013.

Qiang Wang,Hu Zhou and Decheng Wan, "Numerical simulation of wind turbine blade tower interaction," shanghai, 2012.

M. Righi, "Application of computational aeroelasticity to helicopter blades," winterthur.

T. Gornicz, J. Kulczyk, "the assesment of the application of the CFD package OpenFoam to simulating flow around the propeller".

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," IRACT- Engineering Science and Technology an International jornal(ESTIJ), 2011.

Philippe Beaumier, Jean-Mar Bousquet, "Applied CFD for analysing aerodynamic flows around helicopter," 24 th international congress of aeronautical sciences.

Heine, C.J. Meyer and T.w. Backstrom, "CFD simulation of helicopter flow fields," R & D Journal, 2007, 23 (3) of the South African Institution Engineering, p. 8, 2007.