International Journal of Emerging Technologies in Engineering Research (IJETER)
Volume 4, Issue 8, August (2016) www.ijeter.everscience.org

2D CFD Modelling Of H –Darrieus Wind Turbine and Its Optimization for Performance Enhancement

Komal Rawat[1], Hina Akhtar[1], Anirudh Gupta[2], Ravi Kumar[3]
[1] PG Scholar, Department of Mechanical Engineering, BTKIT, Dwarahat, India.
[2] Associate Professor, Department of Mechanical Engineering, BTKIT, Dwarahat, India.
[3] Assistant Professor, Department of Mechanical Engineering, BTKIT, Dwarahat, India.

Abstract – In the present paper 2D CFD model of H-Darrieus Wind Turbine has been developed. The model was implemented in ANSYS Fluent solver to predict wind turbines performance and optimize its geometry. As the RANS Turbulence Modeling plays a strategic role for the prediction of the flow field around wind turbines, different Turbulence Models were tested. The results demonstrate the good capabilities of the Transition SST turbulence model compared to the classical fully turbulent models. The computational domain was structured with a rotating ring mesh and the unsteady solver was used to capture the dynamic stall phenomena and unsteady rotational effects. Both grid and time step were optimized to reach independent solutions. Particularly a high quality 2D mesh was obtained using the ANSYS Meshing tool while a Sliding Mesh Model was used to simulate rotation. .Coefficient of lift and drag were calculated at different values of attack. Main parameter that is monitored in this study is the Tip Speed Ratio (TSR) . Thus, an optimum value of TSR is obtained at which the turbine gives maximum power.

Index Terms – H-Darrieus Wind Turbine; CFD; Transition Turbulence Modeling; URANS; VAWT performance prediction,TSR.

Abbreviations - VAWT (Vertical Axis Wind Turbine),C_l(coefficient of lift) , C_d(coefficient of drag, C_m(coefficient of moment),TSR(Tip Speed Ratio).

1. INTRODUCTION

Vertical Axis Wind Turbines (VAWTs) are becoming ever more important in wind power generation thanks to its compactness and adaptability for domestic installations. However, it is well known that VAWTs have lower efficiency, above all if compared to HAWTs. To improve VAWTs performance, industries and researchers are trying to optimize the design of the rotors. Some numerical codes like Vortex Method or Multiple Stream tube Mode have been developed to predict VAWTs performance and optimize efficiency but they do not provide information on the wakes and they use semi empirical equations to predict effects like tip vortex and dynamic stall.[1]

As it is known, CFD resolves the fluid dynamic equations and it is certainly more realistic than the 1D models but there are many other problematic issues like stall and turbulence modeling, unsteady rotational effects and long computation time. Some works were found in scientific literature [3-8] regarding the application of CFD modeling on VAWTs. The problem in general was the power overestimation due to the arduous prediction of stall phenomena using fully turbulent RANS models for low Reynolds numbers.

Simulations were performed on a AMD processor having 8 cores with 12 GB RAM providing a processing speed of 3.5 GHz. A parallel computing technique was implemented in ANSYS Fluent solver with 6 to 7 processor cores being used at once.

In this paper, the strategy of generating a 2D CFD model to predict H-Darrieus rotors performance and solve such issues is presented.NACA0021 symmetrical airfoil with a three bladed rotor is chosen for the study.

1.1 Working of a H –Darrieus VAWT

When a Darrieus rotor spins, the airfoils move forward through the air in a circular path. Relative to the blade, this oncoming airflow is added in vector to the wind, so that the resultant airflow creates a varying small positive angle (AoA) to the blade. This generates a net force pointing obliquely forward along a certain 'line-of-action', giving a positive torque to the shaft, as shown in Fig. 2. Compared with other VAWTs, Darrieus VAWTs have higher power coefficient. Therefore, Darrieus VAWTs will gradually be used in the modern wind power industry and become the representative of large-scale VAWTs. [2]

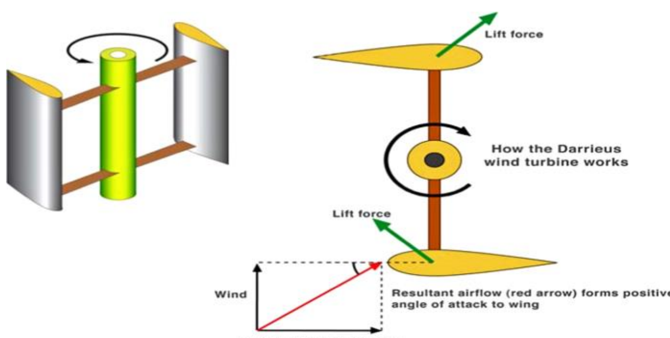


Fig. 1. Operation of Darrieus VAWT.

1.2 Computational Domain Generation and Optimization

Modelling the right Computational Domain is one of the most important task that can be done to simplify the model description or when done in the other way a much simpler or a much complex design can complicate things and make the simulation much more time consuming and costly also at the same time. The mesh can be taken into account as one of the most important parameter in deciding on a mesh independent solution. The advanced turbulence model used in this study requires very fine mesh near the wall region having a Y + value less than 1.

The CAD model was built up in Ansys Design Modeller and out of all the significant setups that were made to simulate the analysis, the best compromise was found for the rotor placed in a rotating ring and bounded by a rectangular fluid domain. The domain has three separated sub domains so as to simulate the desired outcome and so as to use Unsteady Sliding Mesh Model. Figure 9 shows the entire computational domain where three sub domain namely – the rotating ring containing the three airfoil blades is the first, the second is the stationary fluid domain inside the rotating ring and the last being the surrounding fluid medium surrounding the rotating ring in rectangular shape.

A comparison between realizable k-ε , standard k-ω, and SST Transition turbulence model is done which is further used to calculate the mechanical power and power coefficient.

2. CFD NUMERICAL MODEL

The process of generating the 2D CFD model was done inside the ANSYS Workbench multi-physics platform where it is possible to develop a workflow, starting from CAD generation to post-processing of the results. Particularly, the Finite Volume Fluent Solver was used in an Unsteady RANS (URANS) version to solve the Navier-Stokes equations and capture the unsteadiness like the continuous change in the aerodynamic angle of the blade with rotation.

The workflow of the model was as follow:

- Generation of a simplified 2D CAD and computational domain;
- High quality meshing of the domain to meet the specifics of the turbulence models and reach grid independent solutions;
- Setting the Fluent Solver and calibrating the model;
- Optimization of the Transition Turbulence model;
- Post-processing results;

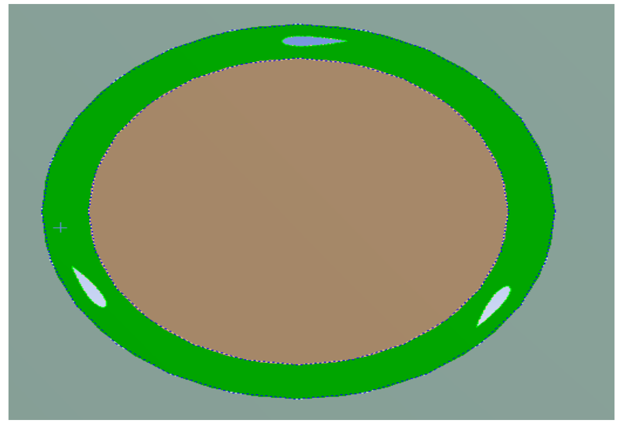


Figure 2 : Rotating Ring Domain

2.1 Mesh Generation and Optimization

Mesh generation was based on generating a high grade spatially discretized grid for the entire domain. The main importance was given to the rotating rings where the most important fluid flow interactions occur. For completing the meshing of the rotating ring, a high quality non-conformal mesh was created using Ansys Meshing. The main meshing tool that was utilized in generating quadrangular mesh around the airfoil walls so as to capture the boundary layer effect required by the turbulence models was the Inflation tool.

The Figure below shows the setup done to create this quadrangular mesh around the airfoil wall so that the Y plus value lies less than 1 around the airfoil walls.

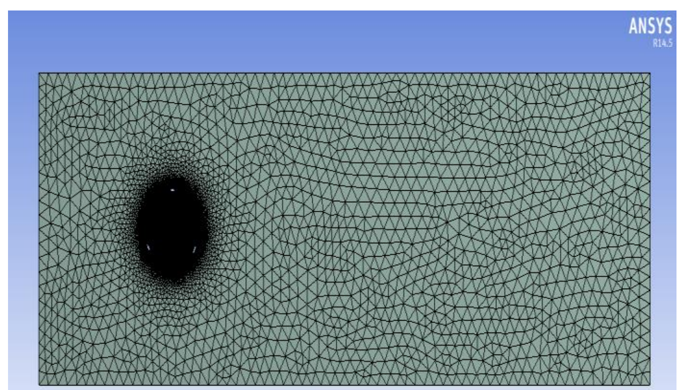


Figure 3: Meshing around the model

2.2Inflation Tool with the required settings

The inflation tool was used for 20 levels of quadrangular cell formation around the airfoil wall with a growth rate of 1.1. The remaining mesh for the entire domain was done using triangular elements having a mesh size.

The final mesh that was obtained after series of test to optimize the results according to mesh size and get a grid independent solution have the following specifications.

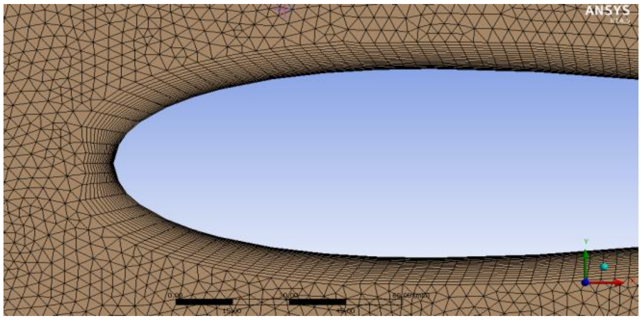


Figure 4: Meshing around the aerofoil

The boundary conditions to activate the Sliding Mesh Model (SMM) were set for the rotating ring cell zone, defining interfaces and rotational speed. In our study, the rotational speed of the rotor is kept constant at 12 rad/s. The BCs for inlet and outlet were fixed by defining designed values for the rotor assembly. Moreover, by following few relevant literatures, the turbulent intensity and turbulent viscosity ratio were fixed at 0.1 percent and 10 for both inlet and outlet.

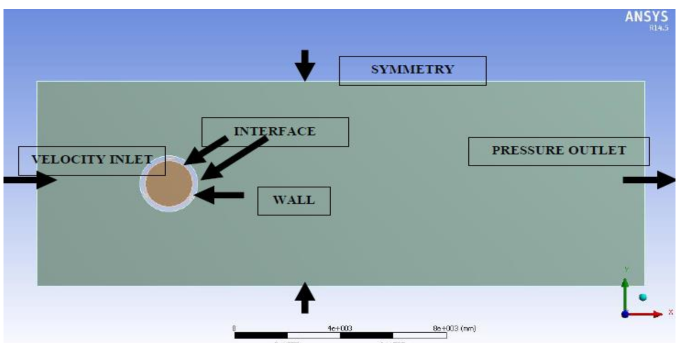


Figure 5:.Computational Domain and BC s

2.3 Solver Settings and Calibration

To take into account the unsteady effects, particularly dynamic stall and interactions between blades motion and wake, it was necessary to use the Fluent solver in a transient version.

The BCs in inlet and outlet were defined fixing the wind speed in inlet at on-design value for each rotor so that the calculated torque is function of rotational speed only and thus it was simple to obtain a Cp versus λ Comparison.

The simulation was done in such a way so that 1 degree of the VAWT rotation can be captured and results may be viewed for the entire VAWT rotation.

In the simulations, the rotational speed of the rotor is kept constant at 12 rad/s or 114.59 rpm.

Thus, N = 114.59 rpm = 1.9098 rps = 687.528 degrees per sec Therefore, 1 degree of rotation = 10^{-3} s and the initial value for the time step is started from this. Again, the maximum number of iterations that were allowed for per time step is 20.

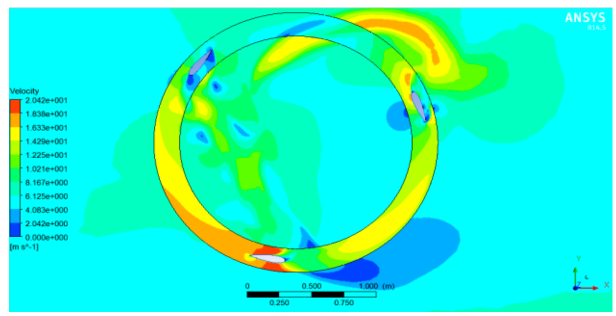
Thus, it is seen that with 10^{-3} time step value and 500 time steps, the total time of rotation for the VAWT is 0.5 sec. Again, since the rotational speed of rotor is 12 rad/s and the rotor radius is 1m. Therefore, time taken to complete one rotation = 0.523 sec.

3. TURBULENT MODEL

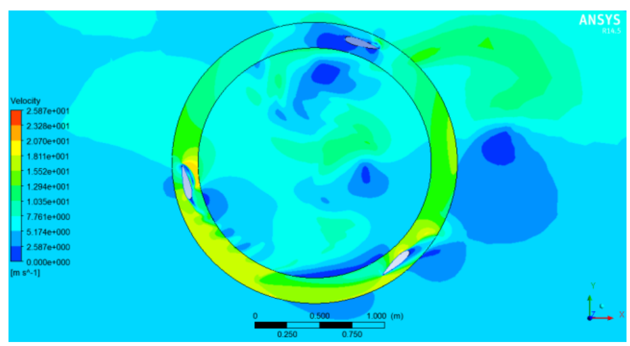
Velocity and Pressure Contours are plotted for 0.1s, 0.2s, 0.3s, 0.4 and 0.5 sec for the three models mentioned below:

1) Realizable k-ε
2) Standard k-ω
3) SST Transition Turbulence Model

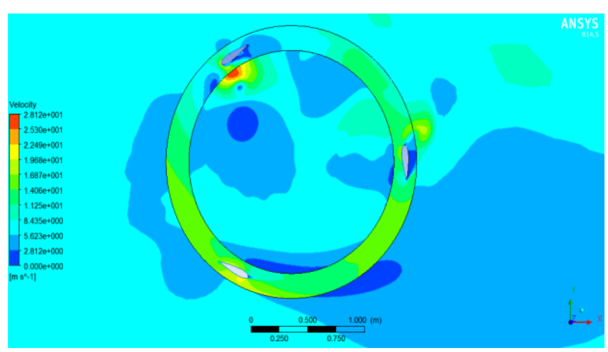
3.1. k-ε Turbulence Model Contours



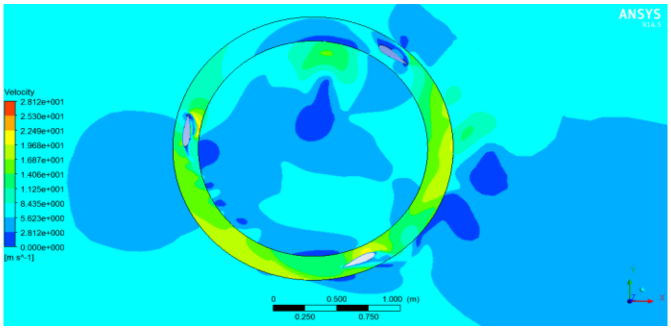
(a)Velocity Contours at 0.1 sec (k-ε)



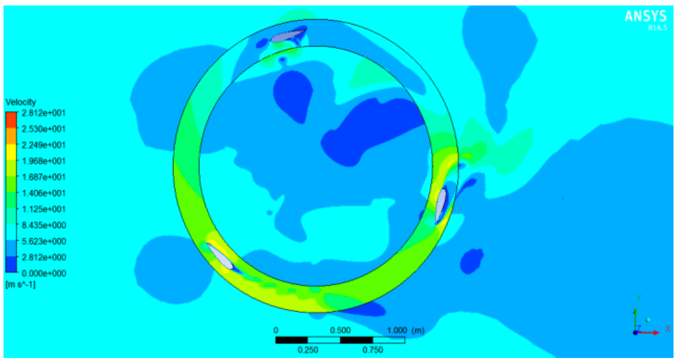
(b)Velocity Contours at 0.2 sec (k-ε)



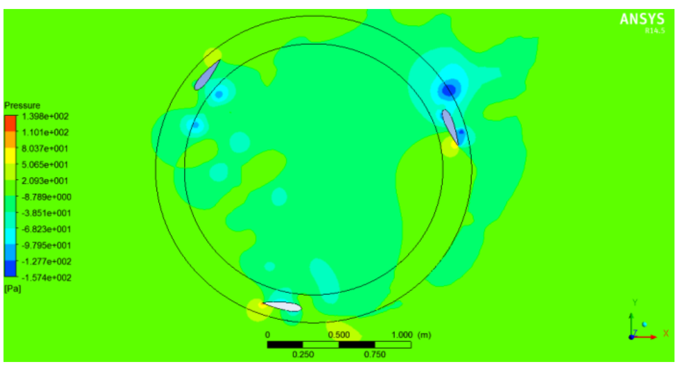
(c)Velocity Contours at 0.3 sec (k-ε)



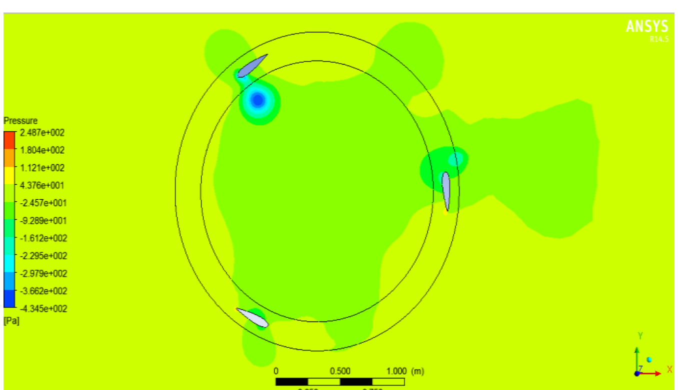
(d)Velocity Contours at 0.4 sec (k-ε)



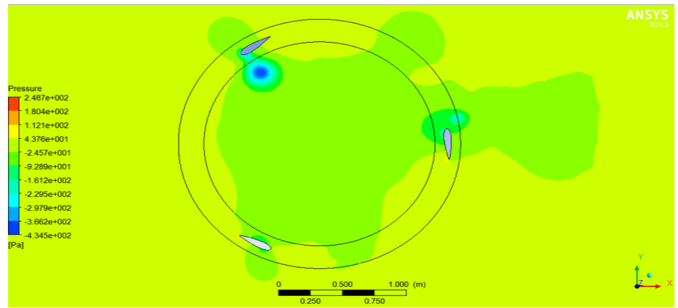
(e)Velocity Contours at 0.5 sec (k-ε)



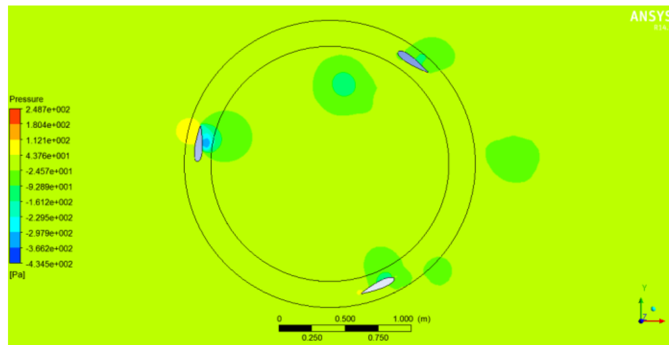
(f)Pressure Contours at 0.1 sec (k-ε)



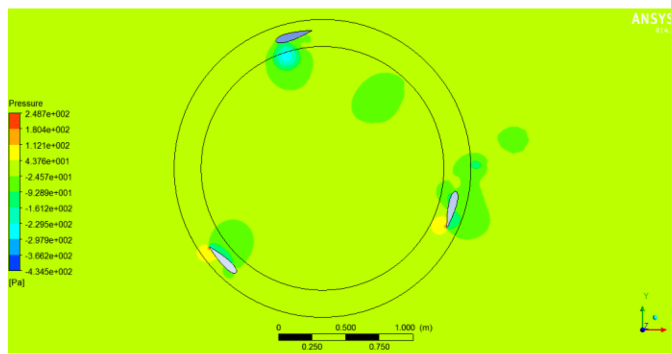
(g)Pressure Contours at 0.2 sec (k-ε)



(h)Pressure Contours at 0.3 sec (k-ε)



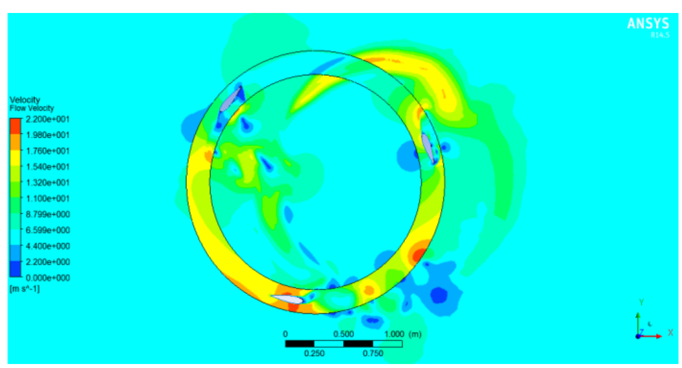
(i)Pressure Contours at 0.4 sec (k-ε)



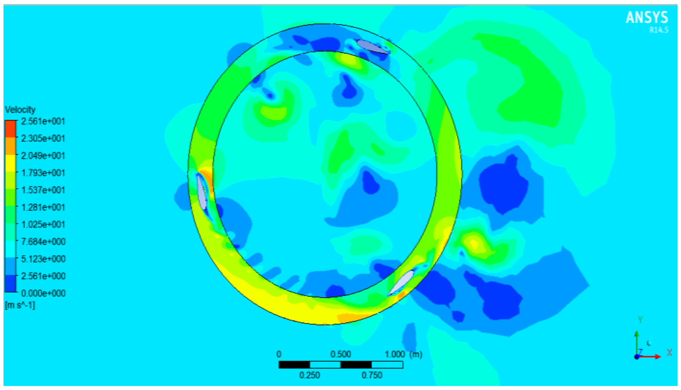
(j)Pressure Contours at 0.5 sec (k-ε)

Fig. 6 : Velocity contours (a to e) and Pressure Contours (f to j)

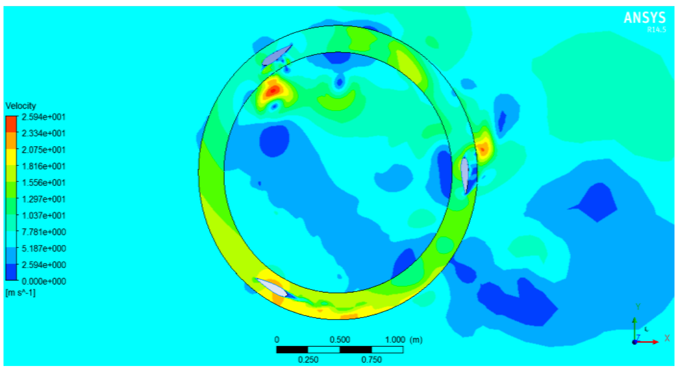
2)k-ω Turbulence Model Contours



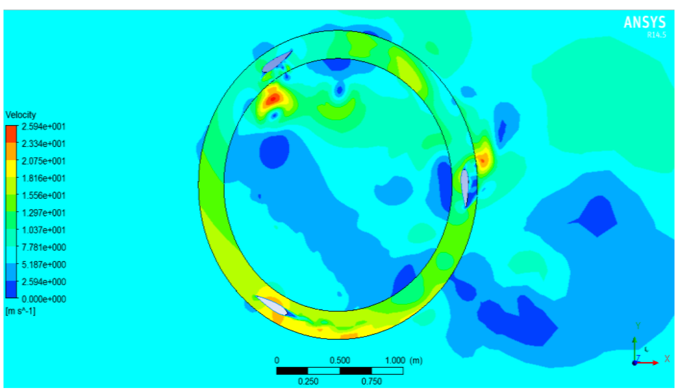
(a)Velocity Contours at 0.1 sec (k-ω)



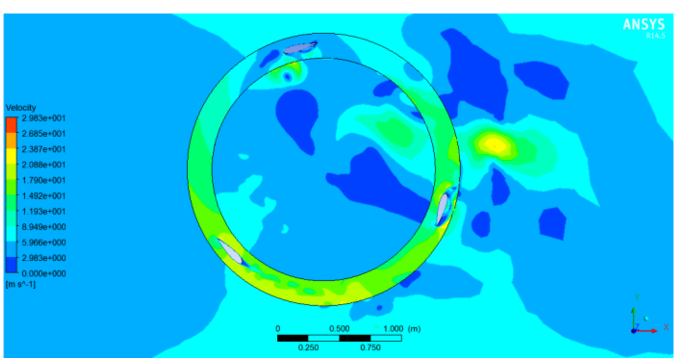
(b) Velocity Contours at 0.2 sec (k-ω)



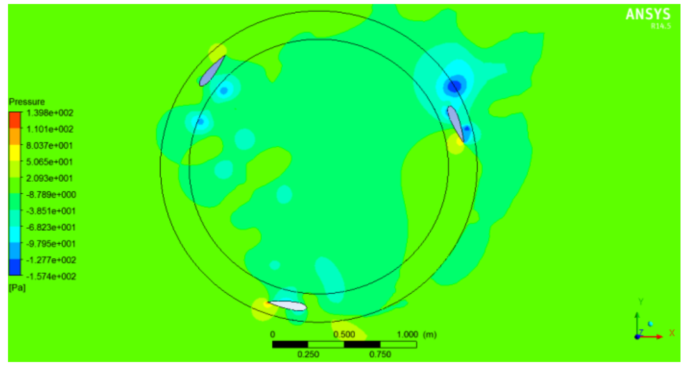
(c)Velocity Contours at 0.3 sec (k-ω)



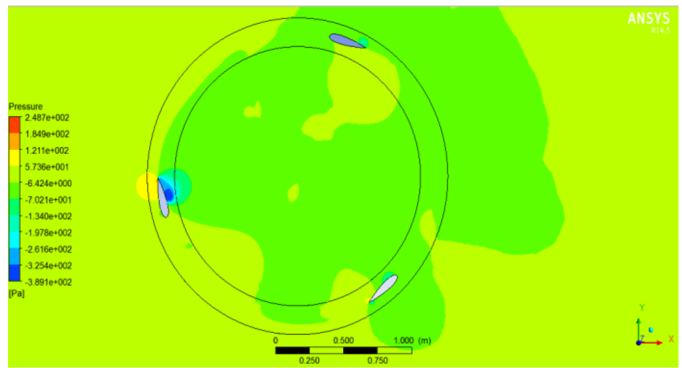
(d)Velocity Contours at 0.4 sec (k-ω)



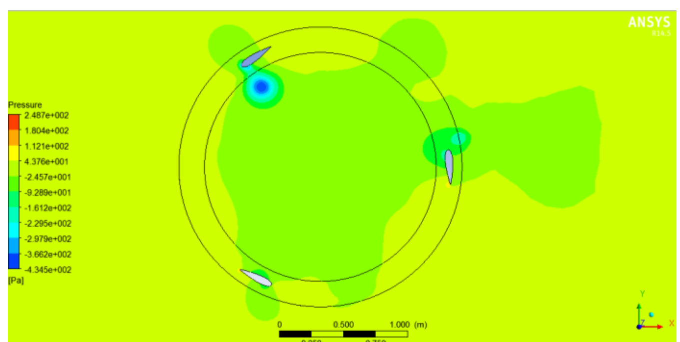
(e) Velocity Contours at 0.5 sec (k-ω)



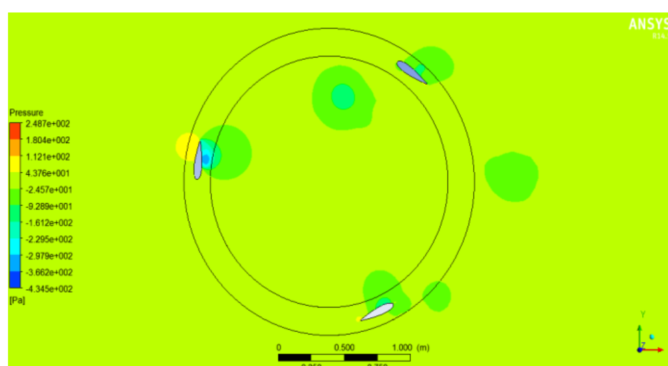
(f)Pressure Contours at 0.1 sec (SST)



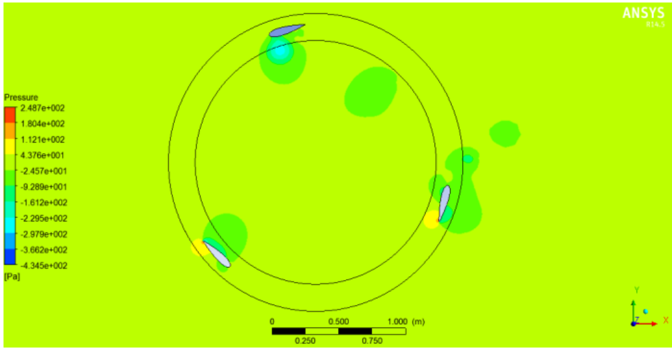
(g)Pressure Contours at 0.2 sec (k-ω)



(h) Pressure Contours at 0.3 sec (k-ω)



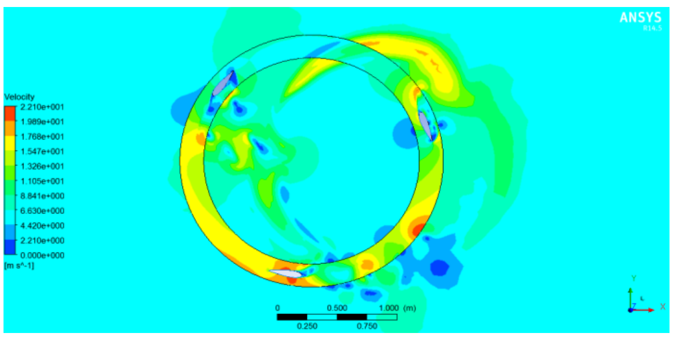
(i)Pressure Contours at 0.4 sec (k-ω)



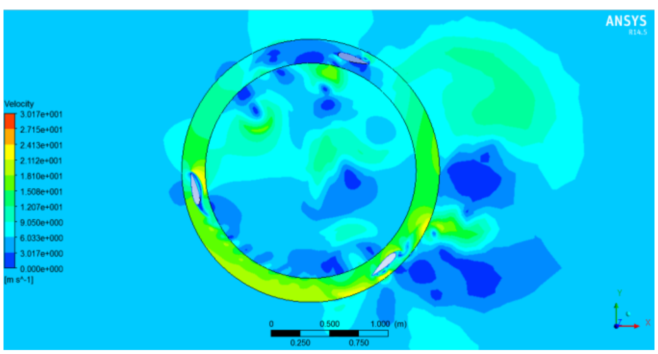
(j) Pressure Contours at 0.5 sec (k-ω)

Fig.7: k-ω model Velocity contours (a to e) and Pressure contours (f to j)

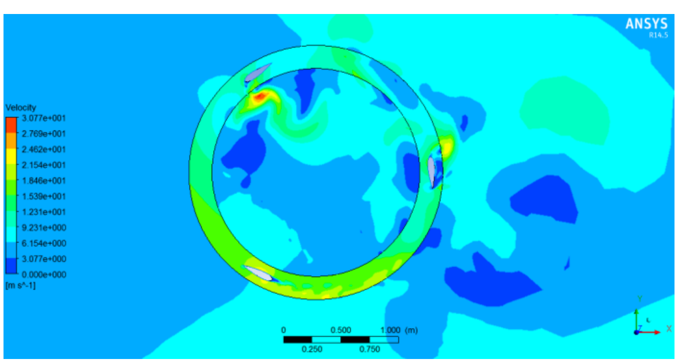
3)SST Turbulence Model Contours



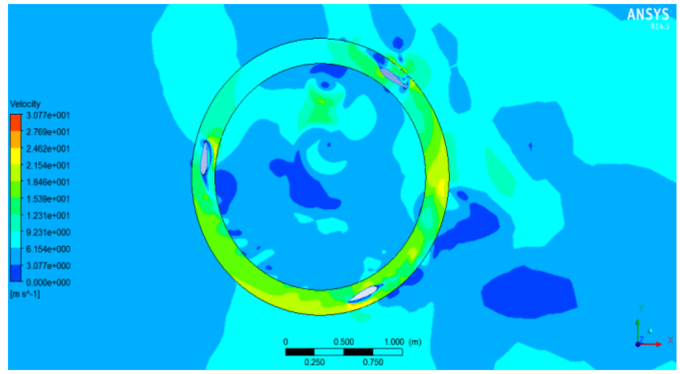
(a)Velocity Contours at 0.1 sec (SST)



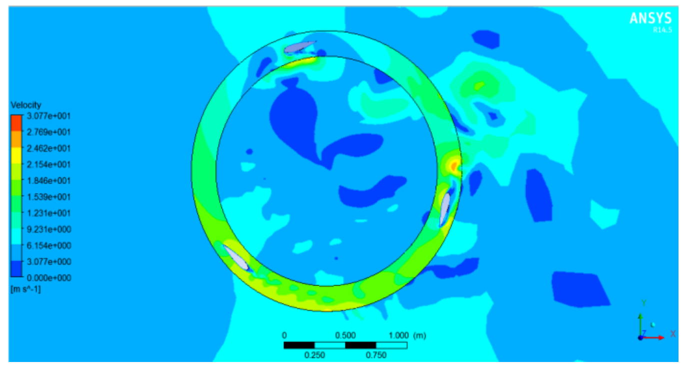
(b)Velocity Contours at 0.2 sec (SST)



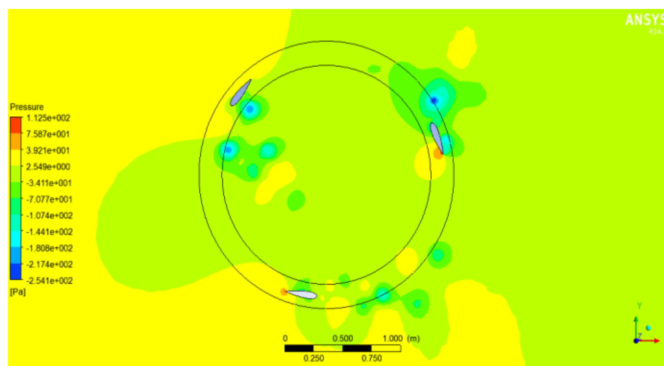
` (c)Velocity Contours at 0.3 sec (SST)



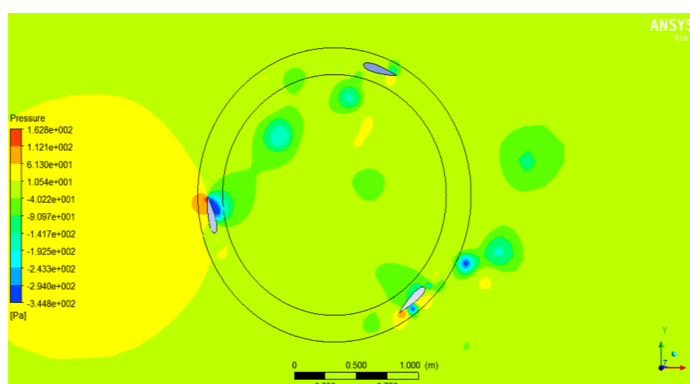
(d) Velocity Contours at 0.4 sec (SST)



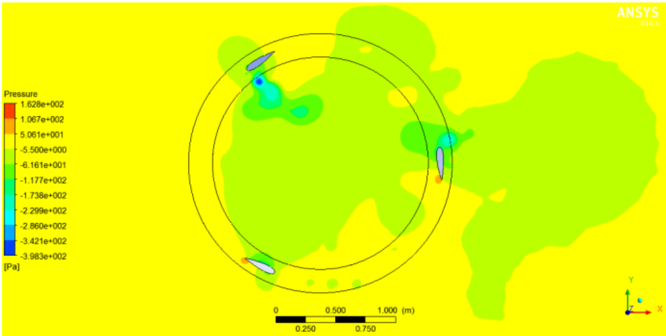
(e)Velocity Contours at 0.5 sec (SST)



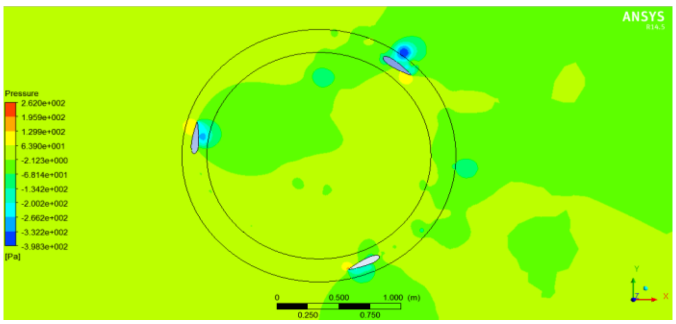
(f)Pressure Contours at 0.1 sec (SST)



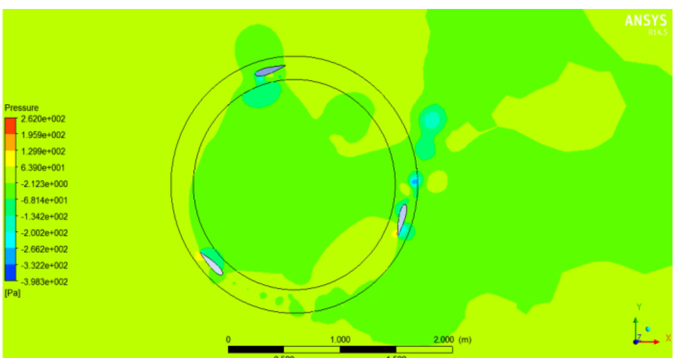
(g) Pressure Contours at 0.2 sec (SST)



(h)Pressure Contours at 0.3 sec (SST)



(i)Pressure Contours at 0.4 sec (SST)



(j)Pressure Contours at 0.5 sec (SST)

Figure 8:SST Model Velocity Contours (a to e) Pressure contours (f to j)

4) Comparative Study

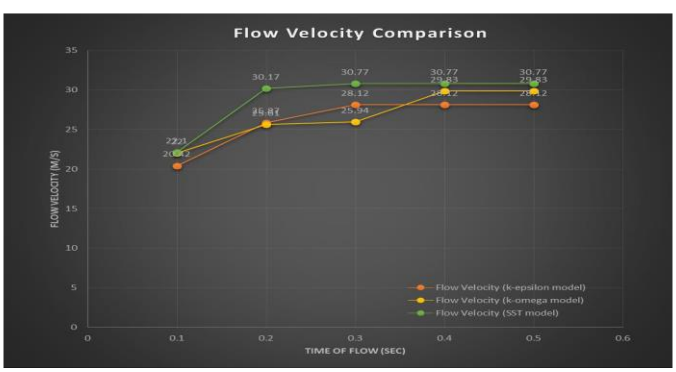


Figure 9.Fluid Domain Velocity Comparison

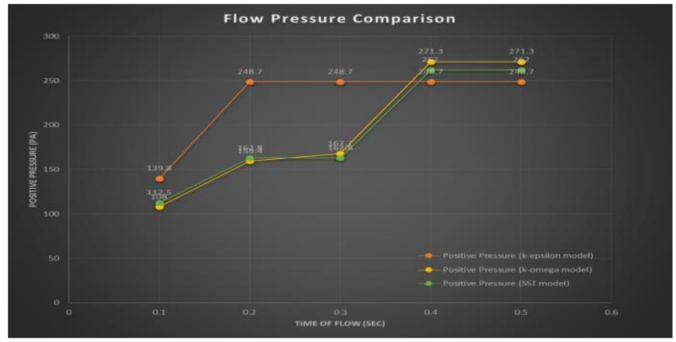


Figure 10: Fluid Domain Velocity Comparison

It is clear that for capturing the velocity field around the airfoil is very delicately captured by the SST model in comparison with the k-ε and k-ω models. There is a steep rise in velocity for every flow advance, however after 0.3 sec of flow continuation, the velocity has reached its limit and the flow can be considered to be highly stable and in a perfect steady condition. However, in terms of positive pressures, both k-ω and SST model show positive results however the k-ε fails considerably in predicting the adverse pressure gradients.

Thus, we can state that SST model gives better results in flow visualization for a VAWT rotor when compared with k-ε and k-ω models.

In order to calibrate the SST Transitional model for wind turbine applications, a long process of optimizing the local correlation variables was carried out.

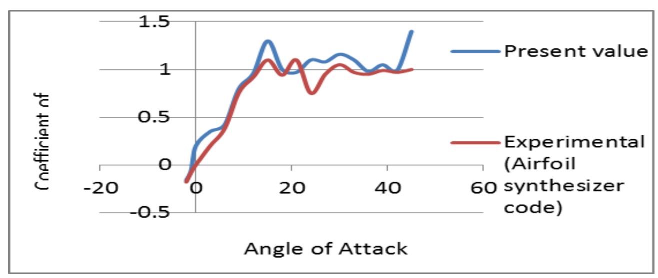


Figure 11: Variation of Lift Coefficient with Angle of Attack for NACA 0021 Airfoil

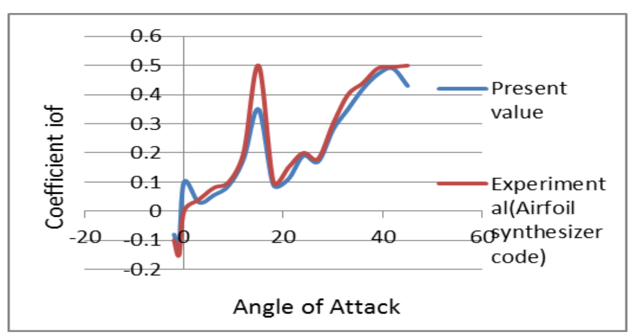


Figure 12: Variation of Drag Coefficient with Angle of Attack for NACA 0021 Airfoil

The coefficient of Drag predicted by the Fluent Solver shows under fitting with respect to the experimental data for most of the angle of attack values and thus makes the assumption pretty complicated. However, the error percentage in this case is also not large and thus the model can be considered stable for these angle of attack values.

4. POST - PROCESSING OF THE RESULTS

Once optimized and calibrated the 2D CFD model, several simulations were performed for the rotor to calculate the power coefficient.

From literature

ω = 2-14 r/s

TSR= 0.2-2.2 (0.2, 0.6, 1, 1.4, 1.8, 2.2)

v= 6.3 -10 m/s

Table 1: Specifications

1	Rotor (R)	1meter
2	Blade Length	1 meter
3	Blade chord	0.2 meter
4	Density	1.225g kg/m^3
5	No of blades	3
6	Wind Speed	6m/sec
7	Rotor Blade Speed	12m/sec
8	Actual rotational speed	12 rad/sec
9	Actual rotational speed	114.59 rpm

Table2: Calculated Cp for different values of TSR

TSR	Radius	ω	V	Cm	T (N/m)	Cp
0.2	1	2	10	0.05	24.5	0.02
0.6	1	4	6.68	0.0376	8.16	0.045
1	1	8	8	0.1105	34.6528	0.221
1.4	1	10	7.14	0.1064	26.575	0.298
1.8	1	12	6.67	0.0928	20.235	0.334
2.2	1	14	6.36	0.0322	6.392	0.142

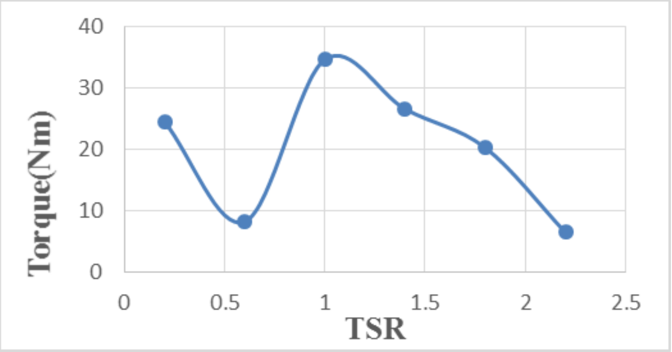


Figure 13 Variation of Torque with respect to Tip Speed Ratio

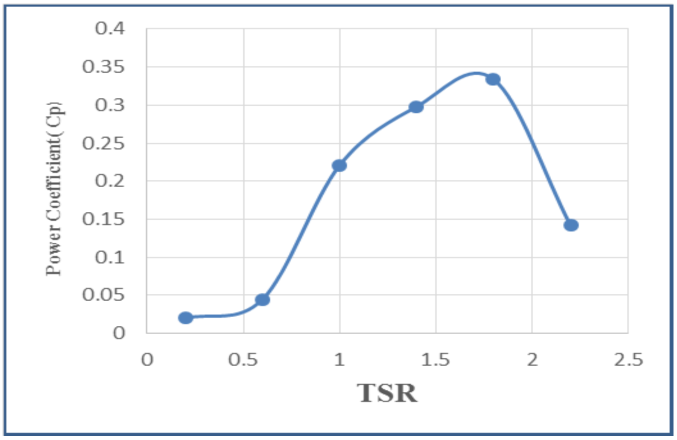


Figure 14 Variation of Power Coefficient wrt TSR

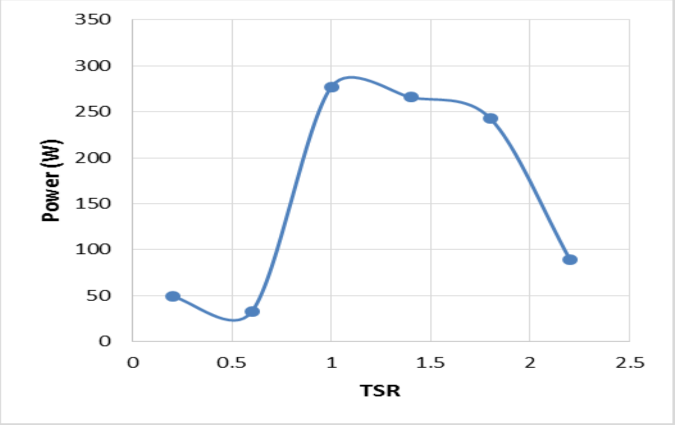
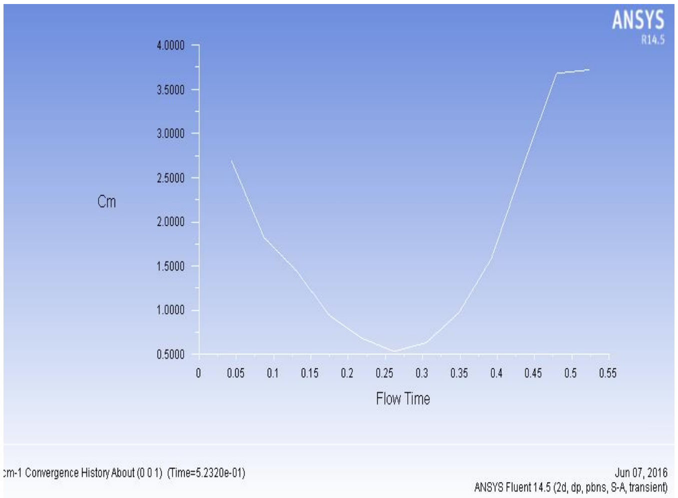
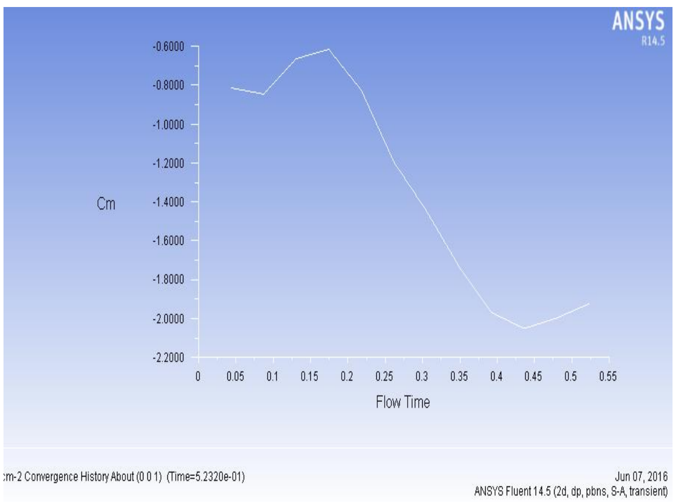


Figure 15 Variation of Power with respect to Tip Speed Ratio

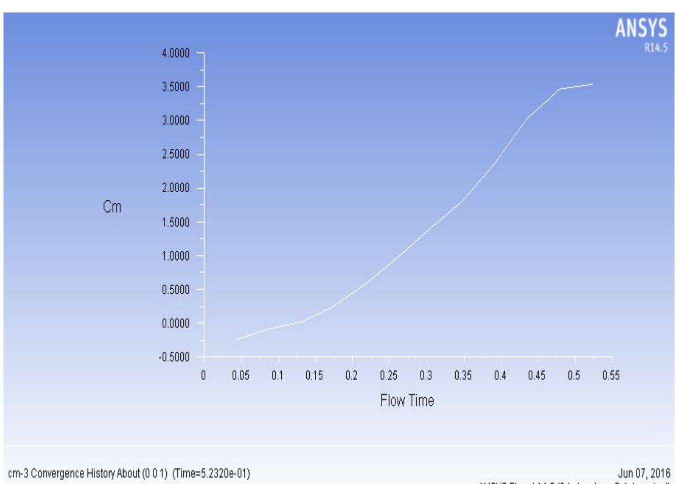
Now the coefficient of moment C_m is calculate for the three aerofoil individually and then a average of all three is plotted with respect to flow time. And it was found that at time 0.523 (which is the time taken to complete one revolution) the C_m is found to be 1.8.



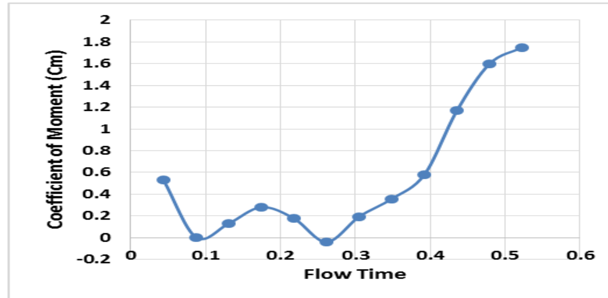
(a)



(b)



(c)



(d)

Figure 16: Coefficient of Moment of (a) Aerofoil 1(b) Aerofoil 2(c) Aerofoil 3(d)Average of three with respect to flow time

5. CONCLUSION

In the present work an unsteady 2D CFD model of H-Darrieus VAWT was developed to evaluate rotor performance and support rotor design and wind tunnel experiments. Computational domain, mesh, solver settings and Transition turbulence model local correlation parameters were optimized. To take into account the unsteady rotational effects, a Sliding Mesh Model was used optimizing the time step of the transient solver. Grid and time step independent solutions were reached after a long process of refinement Coefficient of lift and drag are calculated for the different angle of attack and it was found in close agreement with the experimental.

To optimize the VAWT, coefficient of performance is calculated at different values of TSR and it was found that at TSR=1.8 the C_p was found to be 0.34.

REFERENCES

[1] Lanzafame & etal, 2D CFD Modeling of H-Darrieus Wind Turbines using a Transition Turbulence Model(2014) ,Energy Procedia,Vol 45,pp 131-140.

[2] Jin Xin &etal, Darrieus vertical axis wind turbine: Basic research methods, Renewable and Sustainable Energy Reviews, Vol. 42,pp 212-225.

[3] Mazharul Islam etal (2008), "Aerodynamic models for Darrieus-type straight-bladed vertical axis wind turbines", Renewable and sustainable Energy Reviews, Vol. 12,pp-1087-1109.

[4] Biadgo A. M., Simonovic A., Komarov D., Stupar S. Numerical and Analytical Investigation of Vertical Axis Wind Turbine. FME Transactions (2013) 41, 49-58.

[5] Sabaeifard P.,Razzaghi H., Forouzandeh A. Determination of Vertical Axis Wind Turbines Optimal Configuration through CFD Simulations. 2012 International Conference on Future Environment and Energy IPCBEE vol.28

[6] Guerri, O and etal, "Simulations of the Fluid Flow Around a Rotating Vertical Axis Wind Turbine", Wind Engineering, 2007, Vol. 31, pp. 149-163.

[7] Gupta R., Biswas A. Computational fluid dynamics analysis of a twisted three-bladed H-Darrieus rotor. Journal of Renewable and Sustainable Energy 2, 043111 _2010_

[8] Simao Ferreira C.J., Bijl H., Van Bussel G., Van Kuik G. Simulating dynamic stall in a 2D VAWT: modeling strategy, verification and validation with particle image velocimetry data. The Science of making torque from wind. J Phys Conf Ser 2007;75.