CFD EVALUATION OF HAWT FOR OPERATIONS IN LOW WIND SPEED REGIONS

By

JAWAD AHMAD

Reg # 103698

Session 2016-18

Supervised by

Prof. Dr. ADEEL JAVED

A Thesis Submitted to the US-Pakistan Center for Advanced Studies in Energy in partial fulfillment of the requirements for the degree of

MASTERS of SCIENCE in

ENERGY SYSTEMS ENGINEERING

US-Pakistan Center for Advanced Studies in Energy (USPCAS-E)

National University of Sciences and Technology (NUST)

H-12, Islamabad 44000, Pakistan

JANUARY 2019

CFD EVALUATION OF HAWT FOR OPERATIONS IN LOW WIND SPEED REGIONS

By

JAWAD AHMAD

Reg # 103698

Session 2016-18

Supervised by

Prof. Dr. ADEEL JAVED

A Thesis Submitted to the US-Pakistan Center for Advanced Studies in Energy in partial fulfillment of the requirements for the degree of

MASTERS of SCIENCE in

ENERGY SYSTEMS ENGINEERING

US-Pakistan Center for Advanced Studies in Energy (USPCAS-E)

National University of Sciences and Technology (NUST)

H-12, Islamabad 44000, Pakistan

JANUARY 2019

THESIS ACCEPTANCE CERTIFICATE

Certified that final copy of MS/MPhil thesis written by Mr. JAWAD AHMAD, (Registration No. 103698), of USPCASE (School/College/Institute) has been vetted by undersigned, found complete in all respects as per NUST Statues/Regulations, is within the similarity indices limit and is accepted as partial fulfillment for the award of MS/MPhil degree. It is further certified that necessary amendments as pointed out by GEC members of the scholar have also been incorporated in the said thesis.

Signature: ______________________________

Name of Supervisor: DR. ADEEL JAVED

Date: ________________________

Signature (HoD): ________________________

Date: ____________________________________

Signature (Dean/Principal): __________________

Date: ____________________________________

Certificate

This is to certify that work in this thesis has been carried out by **Mr. JAWAD AHMAD** and completed under my supervision in COMPUTATIONAL laboratory, US-Pakistan Center for Advanced Studies in Energy (USPCAS-E), National University of Sciences and Technology, H-12, Islamabad, Pakistan.

Supervisor: ______________

Dr. ADEEEL JAVED
USPCAS-E
NUST, Islamabad

GEC member # 1: ______________

Dr. EMAD UD DIN
USPCAS-E
NUST, Islamabad

GEC member # 2: ______________

Dr. ADNAN MAQSOOD
USPCAS-E
NUST, Islamabad

GEC member # 3: ______________

Dr. Majid Ali
USPCAS-E
NUST, Islamabad

HoD- (dept) ______________

Principal/ Dean ______________

Dr. Adeel Waqas
USPCAS-E
NUST, Islamabad

Dedicated to my parents and teachers whose tremendous support and cooperation led me to this wonderful accomplishment.

Abstract

Small scale wind turbines can produce power in low wind speed regions and provide them with energy security. In this research, BEMT designed 1kW HAWT is studied for its coefficient of performance and flow field around the turbine using 3D Computational Fluid Mechanics (CFD). Reynolds Averaged Navier Stokes (RANS) equations based numerical simulation is used in this analysis. The turbine blade radius is 1.21 m and the rotor has three blades. To reduce the computational power, only one blade is simulated using the periodicity assumptions in multiple reference frame model. The model is designed in SolidWorks and the grid is generated using POINTWISE mesh tool, employing the T-REX function, mesh independent results are obtained at 10 million elements around single blade. Kω -SST model is employed to model the turbulence. All the simulations are performed in ANSYS CFX package. A detailed selection of results is presented using the ANSYS post-CFD package and TECPLOT 360. The results are validated by comparing the coefficient of performance (Cp) obtained through CFD results and Cp published from BEMT analysis. The Cp of 1 kW rotor from BEMT is 0.48 whereas the Cp obtained from CFD analysis is 0.47, the decrease in Cp through CFD is associated with the rotational losses accounted in the CFD analysis. The simulation results include the Pressure distribution, Velocity distribution along the flow direction at different span ratios, wake behind the rotor, surface streamlines on the turbine blade, turbine power and the torque produced on the rotor blade. The analysis also includes the effect of wind speed on the turbine power.

Key Words: Small Wind Turbine, Computational Fluid Mechanics, Co-efficient of Performance, low wind speed region, ANSYS

Contents

Chapter # 1: Introduction

The power produced through wind technology has greatly increased since last few decades, with global wind power producing above 50GW in 2017 [1], owing to the decline in cost of equipment, encouraging policies, environmental concerns and advancement in wind turbine technology. Wind is an indirect form of renewable energy. The global economy depends upon the availability of cheap and abundant energy resources, where Wind energy provide an environment friendly solution, when depleting fossil fuel resources are an existential threat to the environment and the global economy [2].

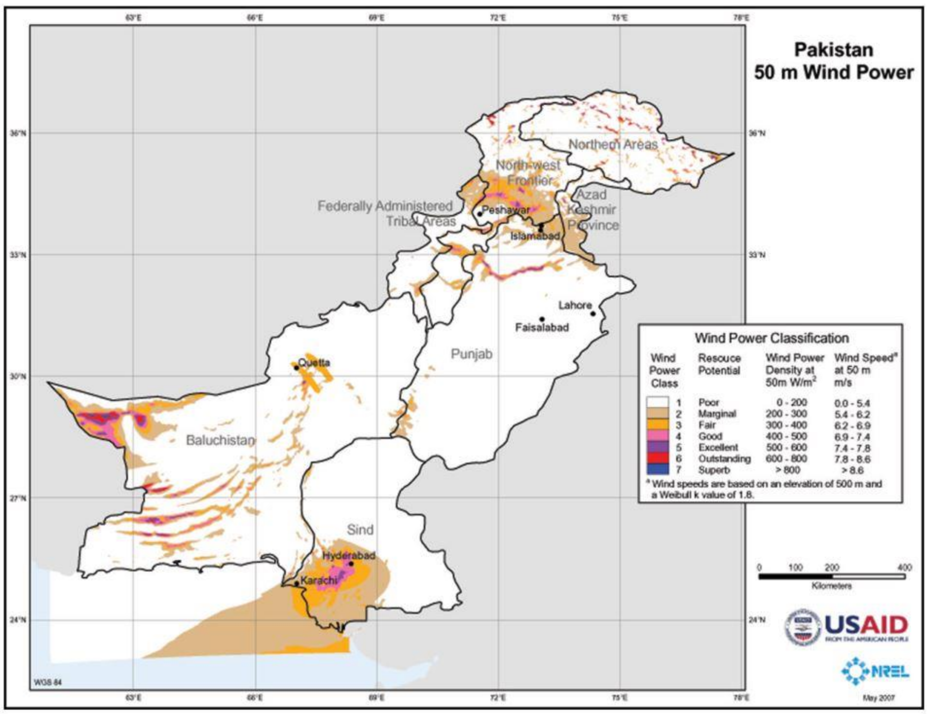


Figure 1: Wind map of Pakistan

The technology to harness energy from wind is improving rapidly, with the development of large rotor blades which could extract considerable energy from wind. Small wind turbines, a type of horizontal axis wind turbines can play huge part in providing clean energy in distributed manner and solution for locations off the grid. A small wind turbine (SWT) with less than 200m^2 of swept area, which makes the radius of rotor blade of 8 m or less and power output equal or less than 50 kW [3]. Also, further small wind turbines are subdivided in three groups based upon their output power i.e. micro 1kW, mid 5 kW, and mini 50 kW [4].

1.1 Energy Efficient buildings and homes

Small wind turbines are an efficient solution for designing a sustainable energy efficient buildings and homes, and the analysis of these turbines have shown to be suitable choice for providing electricity on small scale, as they are suitable for low wind speed regions, can be installed at low heights, urban centers, on or off grid residencies, offshore applications [5]. When Small wind turbines are employed in a distributed arrangement, they become competitive against conventional wind farms. An important part to be examined for use of SWT in Pakistan is to consider their economic analysis for their viability in south Asian climatic conditions. According to the wind mapping (figure 1) shows that urban centers of Sindh, Khyber Pakhtunkhwa and border region of Baluchistan have enough wind potential to support standalone wind systems. One of the economic analysis done at Sindh in the south of Pakistan, showed the estimated cost of electricity produced from wind as 0.0263 US $/kWh. This study shows that installing small wind turbine could help provide the energy needs of an individual home.

This research focuses on finding the aerodynamic efficiency (considering power coefficient and starting time analysis) of a small wind turbine, 1 kW SG6043 airfoil. Wind turbine power coefficient can be analyzed through different aerodynamic models i.e. the panel method, vortex method, blade element momentum theory (BEMT), and through solving of Navier-Stokes equation. Mostly the latter two methods are used, the BEMT is known by its accuracy and fastness, whereas the latter for its accuracy, but it requires high computational power and time. A. Pourrajabian et al. employed the BEMT to design 1 kW horizontal axis wind turbines, considering operating conditions at low wind speed regions. Aerodynamic analysis and optimization of the blade were studied in terms of the two variables i.e. Chord and twist. Apart from power output, starting time of the turbine was included as an objective function in algorithm, considering the wind turbines are designed for installation in low wind speed regions. Both of these goals were optimized by employing a purpose-built genetic algorithm, calculated through BEMT. This paper will use the navier-stokes method to evaluate the aerodynamics of the said blade and compare it with the BEMT method results. The solution to the Navier-Stokes required employing CFD software, which simulates the 3D airflow around the blade.

1.2 Objectives of the Thesis

The objective of study is to validate the BEMT result [pourajjabian], studying the flow analysis around the turbine blade. The analysis is done using the by solving finite volume method and Reynolds's Averaged Navier-Stokes (RANS) equations. The following are the objectives of this study.

1. Designing the 3D turbine CAD model using the cord and twist calculated through BEMT analysis.
2. To reduce the computational resources, periodicity assumptions are used to study only 120^o model.
3. Generating a substantial volume around turbine to study the flow analysis around the wind turbine.
4. Generating mesh and doing mesh independent study to verify the validity of results.
5. Validating the flow analysis by selecting precise turbulence model for this study.
6. Studying the wake and flow around the turbine.
7. Finding out the power produced through the wind turbine at different tip speed ratios and different wind speed.

1.3 References

[1] World Wind Energy Agency (WWEA). Global Wind Energy Report 2017: pg16

[2] G.M.Joselin Herbert et al. (2005). A review of wind energy technologies; Renewable and Sustainable Energy Reviews, Volume 11, Issue 6, August 2007, Pages 117-1145

[3] IEC 61400Wind turbines. Design requirements for small turbines

[4] Wood DH. Recent advances in SWT technology. Wind Eng 189–201

[5] Onder Ozgener (2005), A small wind turbine system (SWTS) application and its performance analysis, Energy Conversion and Management, Volume 47, July 2006, Pages 1326-1337

Chapter # 2: Literature Review

2.1 Analysis of Wind Turbine Aerodynamics through CFD-RANS

CFD –RANS approach was used in this study [1] to analyze a full 3-Dimensional wind turbine rotor using the periodicity assumptions in moving frame reference. The simulations were computed through commercially available finite volume solution Fluent. First objective of the study was to compare the power production computer through CFD with the BEM analyzed power production. Through post processing, study on blade root and blade tip was carried out, so that CFD tool capabilities could be demonstrated a s an optimizing tool. Wake analysis was also carried out.

The BEM designed rotor had 41 m diameter with a 500 kw power output, at a fixed rotational speed of 2.836 rad/s with wind speed of 6.8 m/s. NACA 63-4xx and W3-xxx airfoil were used in outer and inner part of the blade, respectively. Gambit was used to generate the computational grid with 1.5 million volumes, boundary layer was modelled with the toll of wall function.

Fluent, a commercially available solution to compute CFD-RANS equations was used to compute the results and compare with BEM results. The following figure shows the comparison between Fluent and BEM of the computed power and thrust coefficient.

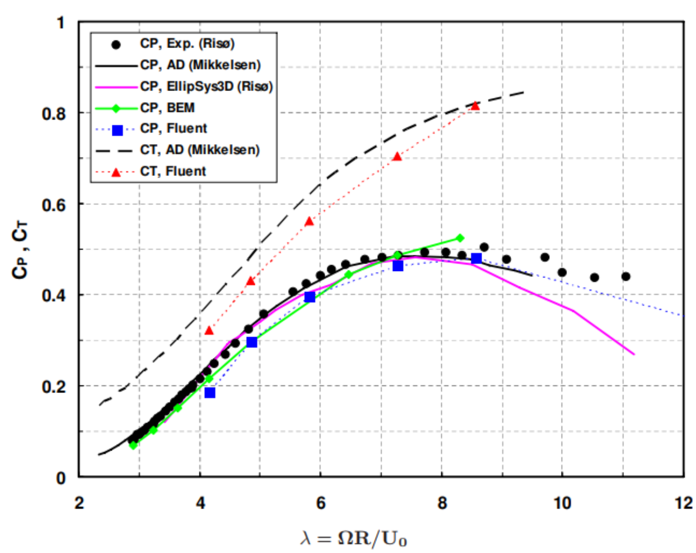


Figure 2: Comparison of Fluent and BEM C_P and C_T

2.2 CFD Analysis of 3D Wind Turbine blade

In this study [2], ANSYS software is validated to be used as a tool for simulations for wind turbine. Through design modeler, mesh and fluent components, the author tries to validate the experimental results obtained through MEXICO blade section at facility of TU Delft. Measurements were performed at standstill board at low speed low turbulence. Velocity of inlet air and fluid flow viscosity was set at 35 m/s and 1.462 kgm/s^2. To solve continuity and momentum navier stokes equations, steady state pressure based solver is used with kω-sst model to solve turbulence. Simple algorithm is used to decouple pressure and velocity. The final comparison shows good agreement between experimental and CFD results.

First step in the CFD validation was to design the blade geometry in design modeler, the next step was to generate a meshing grid. ANSYS meshing tool was to generate the mesh. Local mesh size function was used to refine the mesh. Following parameters were set, Curvature normal to angle at 9^o, blade surface mesh at 0.0025m, with inflation growth rate at 1.2. Around 8 million elements were created. Simulations results were verified by mass flux flow rate. Convergence criteria was set at 10^{-6} and the solution converged after 2000 iterations.

Analysis was done on different turbulence models with Kω-SST gave relatively more accurate results near wall. Also, k-ϵ was find to be not reliable with wind turbine analysis through CFD.

Good agreement has been reached of CFD results with the experimental results for 92% R section but for 60%r section huge differences are recorded. Different turbulent models were also studied which showed kω-sst sowed good results in comparison to k-ϵ and Sparat Allmaras models. The wall distance (y+) was kept to less than 5.

2.3 Using CFD tool to determine downstream wake propagation of Horizontal Axis Wind Turbines

In this study [3], CFD tool was used to determine the downstream wake propagation and the magnitude of velocity at the downstream of wind turbine. There are two areas mainly of importance in downstream region of wind turbine, namely near wake region and far wake region. Near wake is 1D whereas far wake could go up to 20 D, where D is diameter of rotor. Also, in near wake region, Tip vortices effects are important, whereas in far wake region the effect of change in velocity for other wind turbines is of importance. In an unstructured grid, pressure based approach was used on steady flow state for air flowing around the wind turbine, to solve the governing equations employing parallel processing. Also, optimum Wind turbine locations in a wind farm was determined for wind and crosswind directions. The results were comparable to previous findings and confirmed the thesis of this study, that CFD is an excellent tool to study wake properties in wind as well as cross-wind directions in a wind farm. The results were compared to the prevalent wind turbine used in Iran, a 660 kW horizontal axis three bladed wind turbine.

The model was simulated using periodicity assumptions, with only one blade considered, and nacelle and tower neglected. As the wake formation is linear, the domain was selected in a truncated cone shape with 120^o wide angle. The blade was placed as 3D in upstream and 9D in downstream with a total of 12D in Domain.

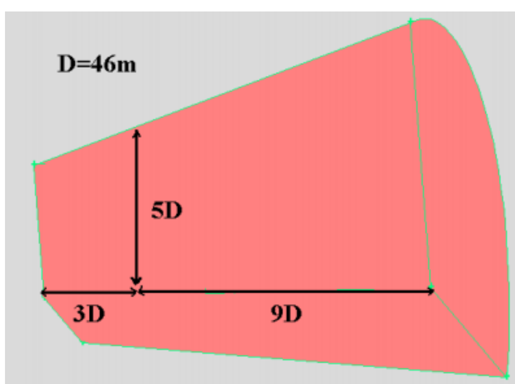


Figure 3: Considered Computational Domain

Multiple reference frame (MRF) capability of fluent was used to consider the effects of the modeled blade, with kω-SST model employed to model the turbulence.

2.4 Design and Analysis of Small Scale Horizontal axis wind turbine for Tainan, Taiwan using BEMT and CFD

In this research, using local Weibull distribution, a small scale horizontal axis wind turbine was designed using BEM Theory and analyzed through CFD. Blade Element Momentum Theory was used to design the shape of the blade whereas aerodynamic efficiency was studied with the help of CFD. The results obtained from BEMT and CFD were both in agreement.

CFD simulations were performed in Fluent package, with kω-SST model. Multiple reference frame was used with periodicity assumptions employed with 120^{o} of the turbine modeled. Grid was made using POINTWISE, with boundary layer region made structured, with the inner volume using unstructured triangular grid, and outer flow region was meshed using tetrahedral cells. The mesh is shown in the following figure 4.

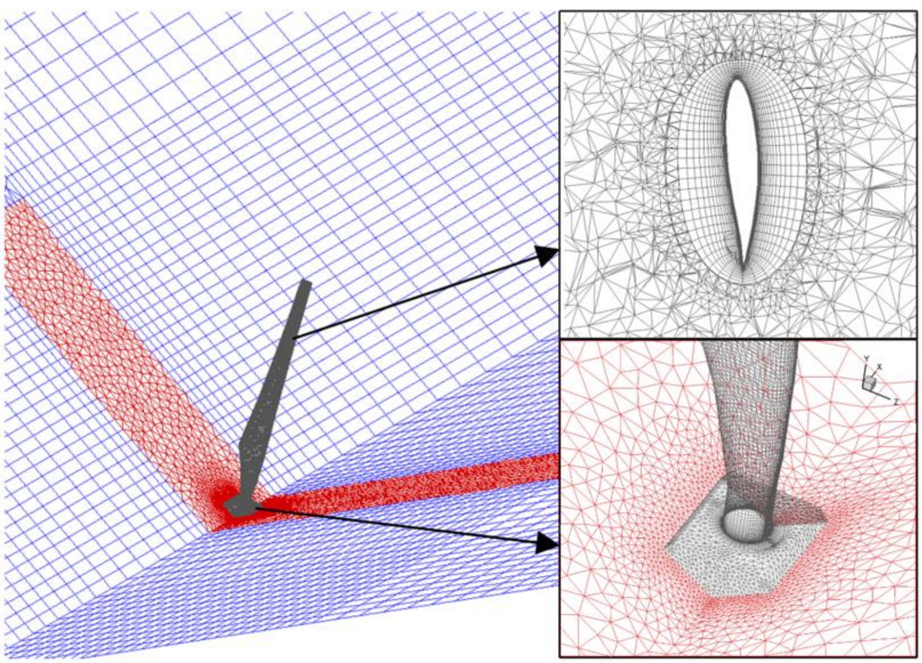


Figure 4: Hybrid Grid in CFD simulation

The grid used was after applying independence test was 1,308,361 cells for CFD simulations.

The highest Coefficient of performance (Cp) was calculated at TSR 6, with BEM showing Cp of 0.430 and CFD evaluating Cp of 0.411. The following graph 5 shows the BEM and CFD curve for Cp at different TSR and Wind speed.

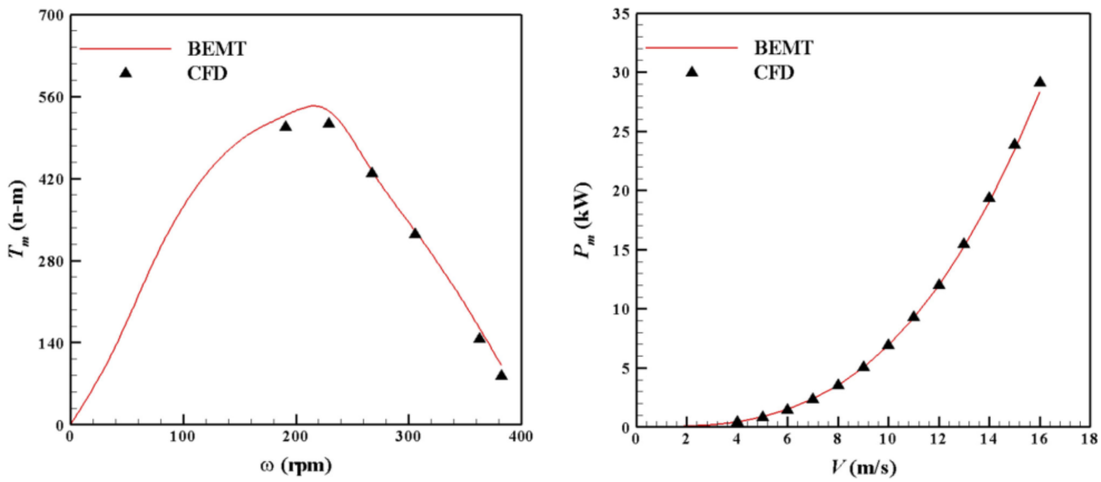


Figure 5 : Torque and power calculated at different TSR and Wind velocity with BEMT and CFD

2.5 References:

[1] Carlo Enrico Carcangiu, CFD RANS Study of Horizontal Axis Wind Turbines. Università degli Studi di Cagliari, 2008

[2] Patrick IMuiruri, Oboetswe Motsamai, Three Dimensional CFD Simulations of A Wind Turbine Blade Section; Validation, Journal of Engineering Science and Technology Review 11 (1) (2018) 138 – 145

[3] Abolfazl Pourrajabian, Reza Ebrahimi, Masoud Mirzaei, Determination of Wake Propagation Downstream of the Wind Turbine using Computational Fluid Dynamics, Proceedings of ASME Turbo Expo 2012, June 11-15, 2012

[4] Chi-Jeng Bai, Po-Wei Chen, Wei-Cheng Wang, Aerodynamic design and analysis of a 10 kW horizontal-axis wind turbine for Tainan, Taiwan, Clean Techn Environ Policy (2016)

CHAPTER # 3: CFD Modelling

3.1 3D Rotor CAD Design

The CFD analysis of the 1 kW HAWT started from its geometry. To reduce the computational requirement only one blade was analyzed, taking advantage of the 120o periodic assumptions. To achieve the results presented in [8], only the blade part was considered. A 3D SolidWorks blade model was created using the geometrical data for the 1 kW small wind turbine. As the turbine was optimized considering two objective functions, power production and starting time, the weightage factor of w = 0.7 was considered as it decreased the starting time by 29.7% whereas Cp was decreased only by 3.2% [1]. SG 6043 airfoil was used along blade, which has a high lift to drag ratio [14].

In BEM analysis, the turbine blade was partitioned into fifteen sections, therefore 15 planes were created in SolidWorks. Since the BEM produced scattered distribution for chord and twist, which could not be used for manufacturing, they were smoothened by third order polynomial fitting [15]. Wood found that fitting least square polynomials to chord and twist distribution led to smooth curves. The chord length near the root are larger as compared to the tip, also the twist is increased near the root, since they have greater effect on starting time of turbine.

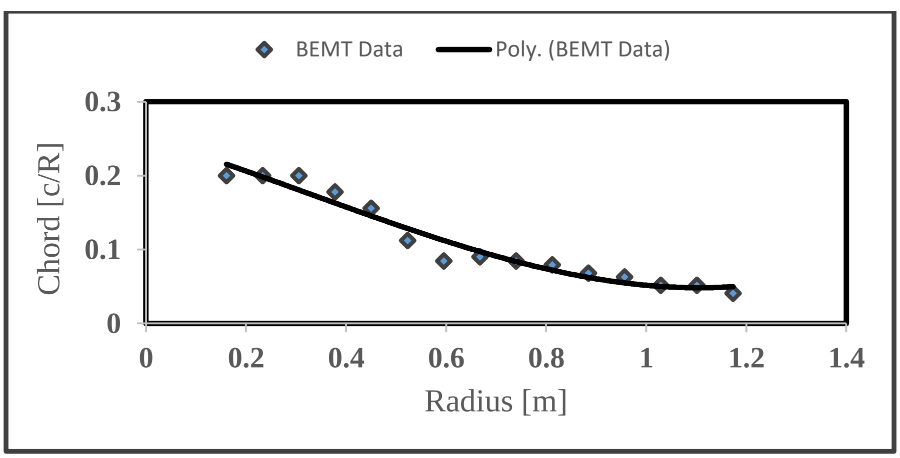


Figure 6: Chord fitting for CAD model

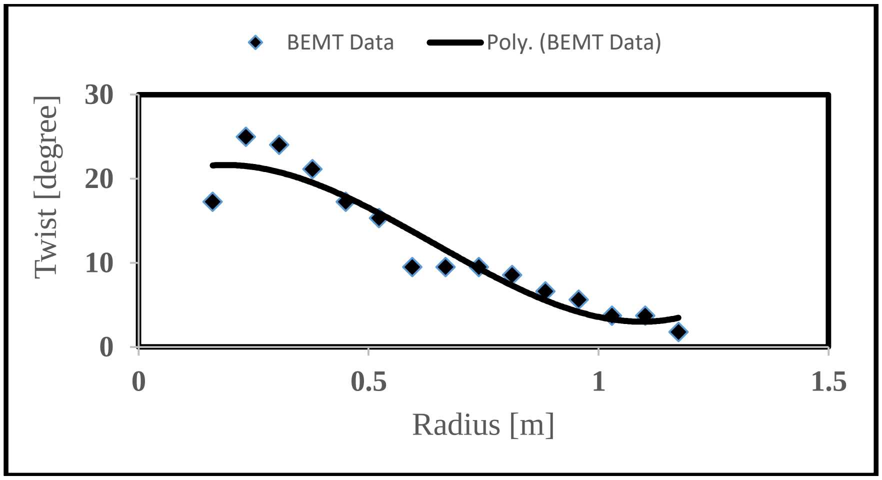


Figure 7: Twist fitting for CAD model

Table 1 shows the tabulated data of chord and twist used along the blade length. The blade hub length is 0.125 m and the total length of the blade is 1.21 m, the first and last plane chord and twist were extrapolated. The blade length was divided into 15 sections, equally spaced in span wise direction.

Plane #	***Radius***	***Chord [c/R]***	***Twist [º]***
1	0.125	0.224	21.348
2	0.161	0.215	21.583
3	0.233	0.198	21.501
4	0.305	0.180	20.778
5	0.378	0.162	19.527
6	0.450	0.145	17.861
7	0.522	0.128	15.896
8	0.595	0.112	13.744
9	0.667	0.097	11.520
10	0.739	0.083	9.336
11	0.812	0.072	7.308
12	0.884	0.062	5.547
13	0.956	0.054	4.169
14	1.029	0.050	3.287
15	1.101	0.048	3.014
16	1.173	0.049	3.465
17	1.210	0.051	3.997

Table 1: Chord and Twist data along the blade

Parameters	***Values***	***Units***
Rated Power P_{rated}	1	kW
Tip speed ratio	5.71	-
Number of Blades	3	-
Angle of attack	5	(^{o})
Rated wind speed	10	m/s
Rated rotor speed	450	rpm
Rotor radius	1.21	m

Table 2: Main parameters of 1 kW small horizontal axis wind turbine

Table 2 shows the parameters of the 1 kW HAWT designed and analyzed through BEMT and CFD, respectively. Number of blades are set to be 3, although increasing the number of blades in small wind turbines increase the power performance but in such scenarios a comprehensive economic evaluation needs to carried out. Most of the parameters were optimized using empirical relations [2] .

3.2 Computational Fluid Dynamic Methodology

The following is the flow diagram which will be followed for running a successful CFD simulation. Each part will be explained separately in next sections. The CFD will have three portions, namely pre-processor, solver and post processor, whereas in validation the CFD results would be compared to BEMT results given by Pourrajabian et al.

Co-efficient of performance calculated from BEMT and CFD are compared in Chapter 4. The mesh generation, one of the most tedious task of CFD analysis is done with the help of POINTWISE®. The convergence criteria are also set to an optimum of 10^{-6} to achieve better convergence and saving time as well. Also, the post CFD results are evaluated with the help of ANSYS post-CFD and using TECPLOT ®.

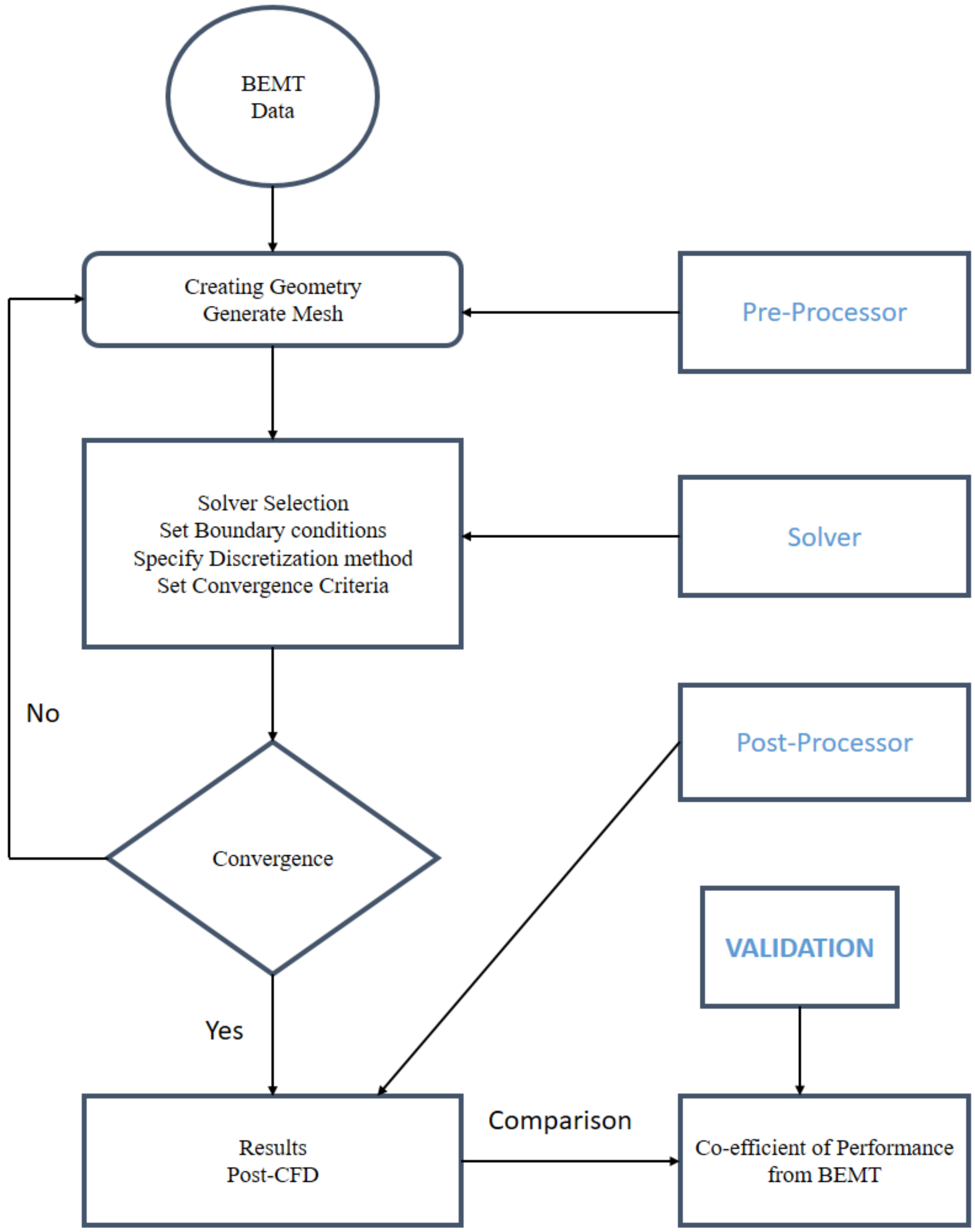


Figure 8: Methodology Flow Chart

3.3 CFD Domain

Only one blade of the turbine was simulated using the periodicity assumptions, as the turbine is symmetrical around its axis of rotation. The benefit of using periodicity assumptions is that it helps saving time and resources. The computational domain is created in Design modeler. It is shown in following figure.

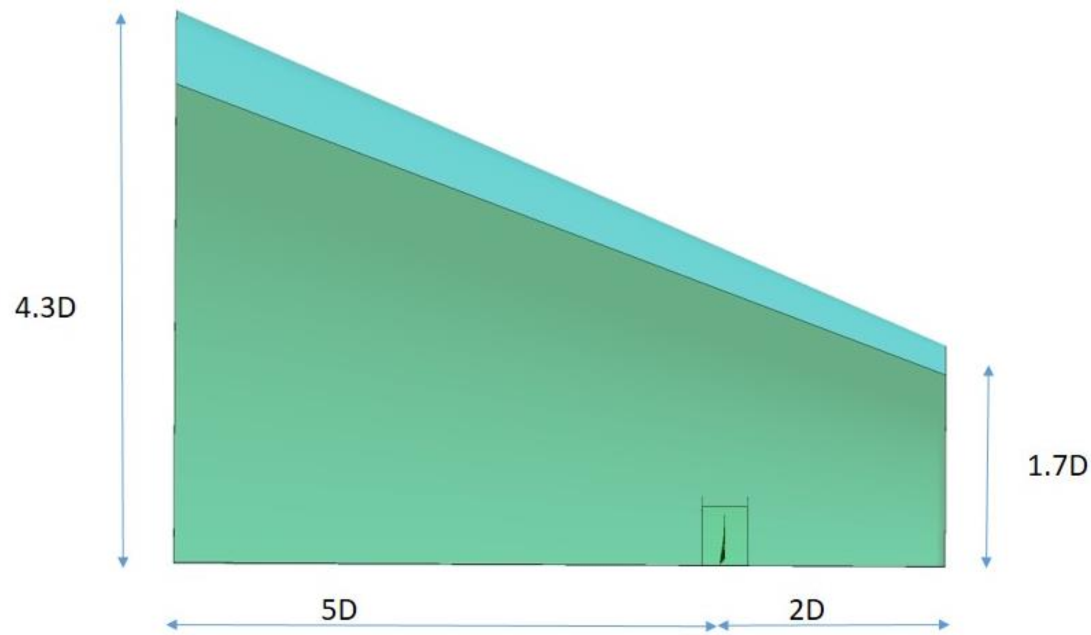


Figure 9: CFD Domain

A conical shaped domain was created, as it has been reported in prior research [1], that wake expansion is better studied in conical expansion domain. The Domain is dimensioned using the diameter, D of the turbine. The inlet is created using 1.7D whereas the outlet is of 4.3D, to make a truncated cone shape. The turbine blade is placed at 2D from the inlet, whereas to study the wake effects on the blade downstream of blade is of 5D.

A small volume has been enclosed in the main domain to study to capture the boundary layer study the effects of viscosity on the turbine blade. Blade is rotated using multiple reference frame (MRF) feature of ANSYS. In this module steady state conditions are observed at the interface of the two intersecting volumes and absolute velocity is considered to be same for each frame reference [3].

3.4 Boundary Conditions

Boundary conditions is one of the most important part of the CFD simulation, as it is our gateway to tell the CFD algorithm the conditions at different sections of the domain, and what are these sections to be considered and what values are to be assigned to them. The following are different sections considered in this simulation.

3.4.1 Blade

Blade was considered as wall (no-slip condition). It was considered as a solid, on which fluid experiences zero acceleration/velocity and creates shear stress cross the fluid across the domain, and creates turbulence, which needs to solved according to some created turbulence models. there are several turbulence models but as explained in Chapter 2, most of the researchers are of the view that kω-sst model is very accurate in modeling Horizontal axis wind turbines turbulence, which is the same used in this study.

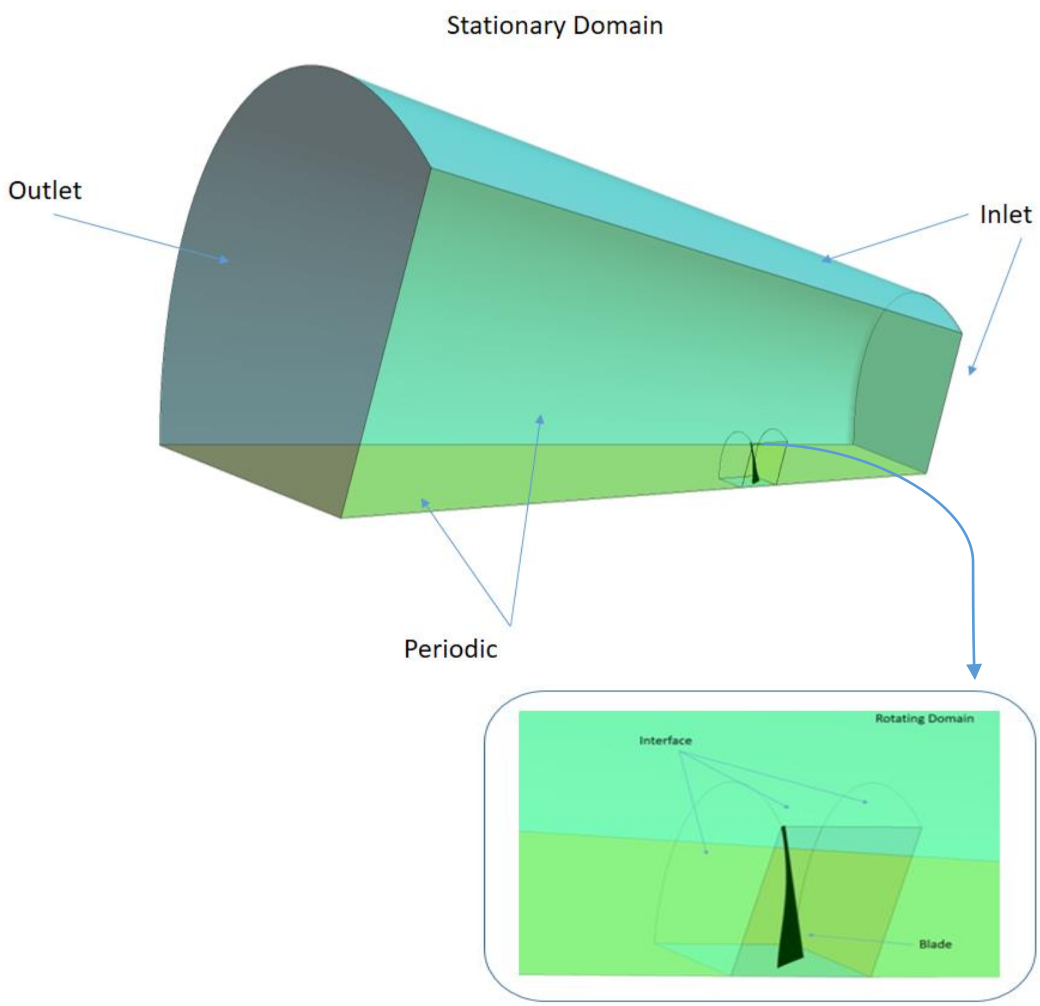


Figure 10: Domain Boundary conditions

3.4.2 Inlet

As shown in figure 10, inlet is specified at two sections of the domain, one is the axial inlet whereas the other section created due to the cone geometry. Inlet is a velocity inlet condition, with design inlet velocity of 10 m/s, and changed accordingly for subsequent simulations to analyze wind turbine behavior at different wind velocities.

3.4.3 Outlet

Outlet is specified as a pressure outlet, downstream of 5D from the blade. It is taken as atmospheric pressure. It is an approximation to reduce the computational resources needed to solve at a lower downstream, as the results from the Post-CFD shows that due to change in pressure around the turbine, the pressure is not exactly at atmospheric pressure. But this approximation does not have much impact on our study on Co-efficient of performance and impact of wind velocity.

3.4.4 Periodic

Periodic boundary condition is used when the system has flow characteristics which are periodically repeating across the system. As the designed turbine is of three blades, where each blade is identical the fluid flow across each of them is identical. Therefore, by applying periodic conditions we can save time and resources to simulate the whole turbine.

3.5 Grid Development

Grid generation is one of the most tedious task of a reliable CFD simulation, as the accuracy of results are directly related to fine generation of mesh. Irrespective of solver setup, mesh plays a direct role in analysis of the results. There are few methods to develop fine mesh, which are explained below.

3.5.1 Structured Grid

Structure grids have a great degree of similar patterns, called blocks, which are followed across the whole domain, so the solver is able to transmit the results across the blocks with high accuracy and precision. These grids mostly utilize hexahedral elements. And they have high orthogonality factor with a very less skewness factor.

Advantages of using structured grid is it saves computing times, due to its high degree of structured blocks, thus consuming comparatively less memory size in computing results. It also allows a high degree of control across the domain.

Whereas disadvantages include, a very time consuming process of building the grid, and in a 3D setup, it becomes very tedious task. And sometimes near to impossible owing to structure of 3D turbine blades.

3.5.2 Unstructured Grid

These type of grids utilize triangles and tetrahedral to fill the domain, which is done in an unstructured pattern, with little to none user input required.

Advantages of unstructured domain is its employment in 3D geometry which have high degree of curves and thickness, and grid could be assembled in less time, with less input from user.

Whereas, the disadvantages of such grids, since it fetches little input from user, the CAD geometry domain could experience errors like negative areas, mostly in 3D setting. Also, it requires a higher memory size and relatively higher time to solve the equations.

3.5.3 Hybrid Grid

There are always certain parts of the domain which are of more importance to the researcher than other parts of the domain. So in an Hybrid Grid, The portions which are more importance are structured grid, whereas other portions are unstructured.

Advantage of using such a domain, is it gives an optimum result, by giving accurate result, whereas also, saving time. But it requires a higher degree of literature review, so all the parts of the domain which are of high importance are significantly given fine mesh.

In this study, power performance was of importance, therefore, boundary layer region and wake needs to fine meshed, whereas all other could employ unstructured grid.

3.5.4 Domain

This wind turbine domain does not include tower as well as HUB. As explained earlier, more time was spent on creating fine mesh around the turbine blade and the boundary layer, as will be explained in next chapter, reduction of identifying the torque by 10^{-1} incorrectly, gives reduced to Cp, which does not allow us to validate the BEMT results.

Following figure shows the Grid around the domain.

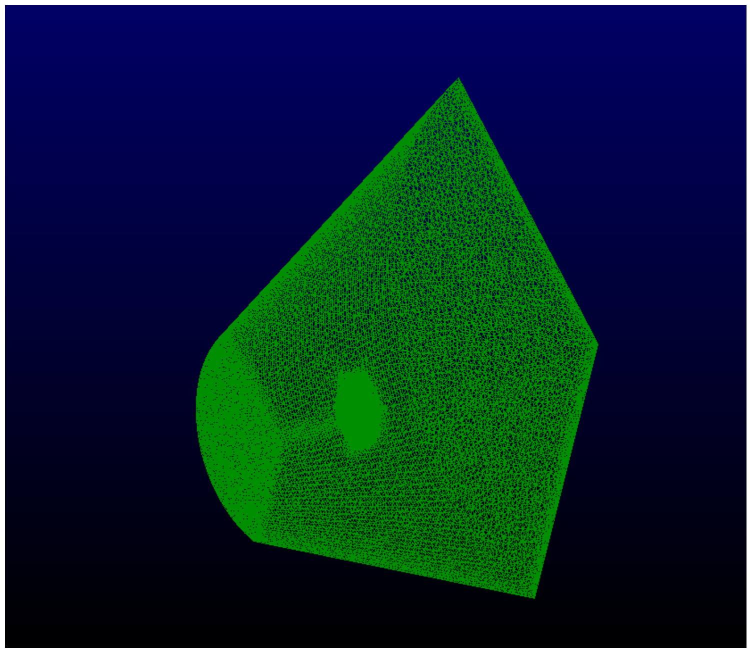


Figure 11: Grid Domain

- Stationary Domain: all the surface meshes are unstructured. The two bottom surfaces are built with the periodic function, to grant the necessary symmetry for the solver periodicity conditions. The volume mesh is 1.5 million tetrahedral elements.
- Rotating domain: surfaces and volume were meshed as in the outer domain. Cell size reduces while approaching to boundary layer of the blade. The volume contains 8.2 million elements tetrahedral volume elements and 2.3 million pyramids volume elements.
- Blade surface: structured mesh was adopted for all the blade surfaces, except for the tip, where an unstructured mesh was used. After some sensibility tests and accordingly to previous works, 314 nodes were positioned around the airfoils, 157

for each side. To avoid the formation of very skewed volume elements near the trailing edge region, but keeping at the same time a low total cell number, 314 nodes were positioned on the whole blade in spanwise direction. The most problematic zone from this point of view was the highly twisted transition section.

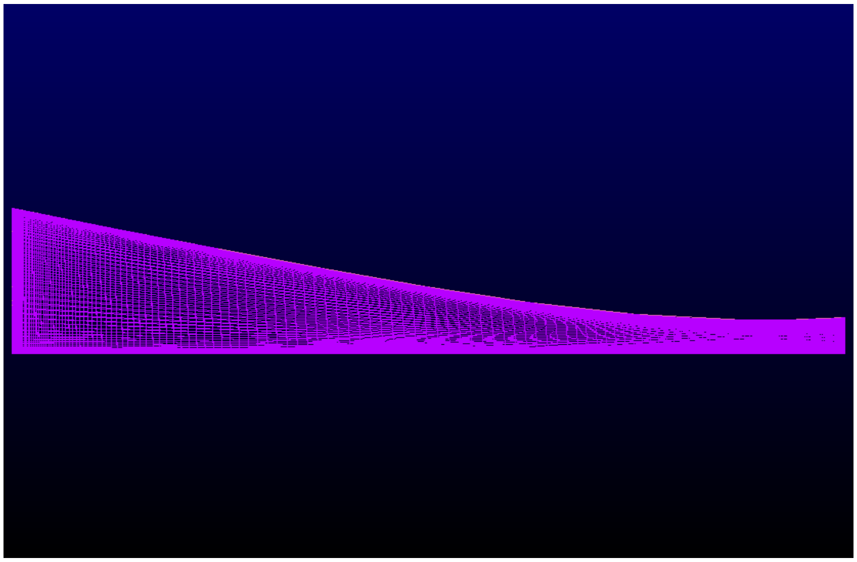


Figure 12: Structured Mesh on Suction side of blade

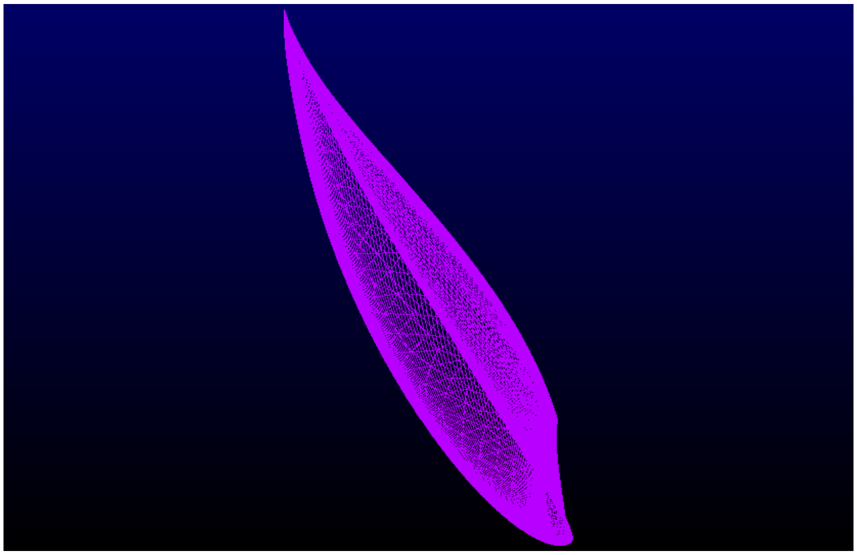


Figure 13: Unstructured Mesh on Blade hub connection

- Boundary layer: the blade's boundary layer consists of 1.3 million mixed hexahedral and prismatic elements, being the latter over the tip surface. The T-Rex function was applied to the blade surface, with 30 steps and a 1.2 grow factor.

- The height of the first boundary layer cell was tuned in order to keep y+ < 10 over the entire blade surface for all the examined wind speeds. This ensures the coherent application of the $k - \omega$ SST turbulence model.

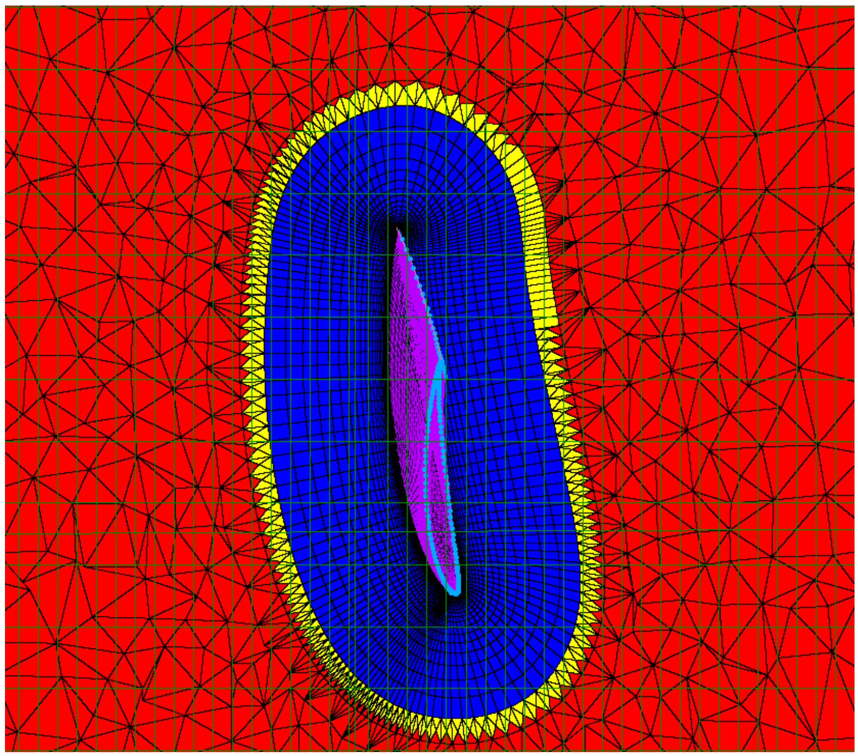


Figure 14: T-Rex Function layers on Boundary layer

Pointwise allows also to initialize boundary and volume conditions for various solvers, and in this case ANSYS CFX was selected. Each spanwise section was divided into pressure side, suction side and trailing edge wall conditions. Blade, inlet and outlet were set to wall, velocity inlet and pressure outlet.

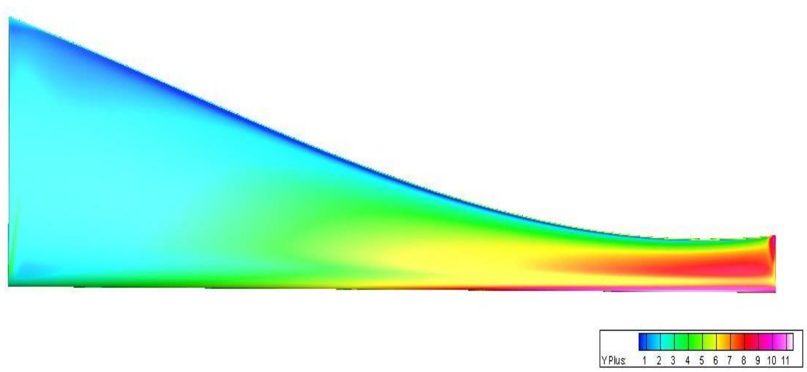


Figure 15: y+ at Suction side of blade

Since in this study, near wall modelling is of high importance, therefore we need to keep a low Y plus. It is recommended to maintain a Y plus of less than 10 [4]. Y plus is a non-dimensional factor studied for a wall bounded flow.

$$y^{+} \equiv \frac{u_{*}\, y}{\nu}$$

'U_*' is friction velocity, whereas 'v' is the local kinematic viscosity and 'y' is the near wall distance.

3.6 Grid Independence Test

Grid independence is one of the few test which can predict about the reliability of the results. Grid independence means that the calculation results does not change very by so small quantity, that such small error could be ignored in such study. This test is an analogous to accuracy of results and shows the rationality of numerical simulation. It is a good practice to increase the cell size by 1/3rd and then compare the results of neighboring simulations, if the results of both simulations are near identical, then the grid could be said to be grid independent [5]. Theoretically, the CFD solution completed through grid independent results, should give identical solution to governing equations. This criterion could be used to validate different numerical and mathematical models [6].

Coefficient of performance calculation and comparison with BEMT result is one of the objectives therefore, grid independence test was done with torque produced with number of cells in the grid, so increasing the number of cells have no change in power produced.

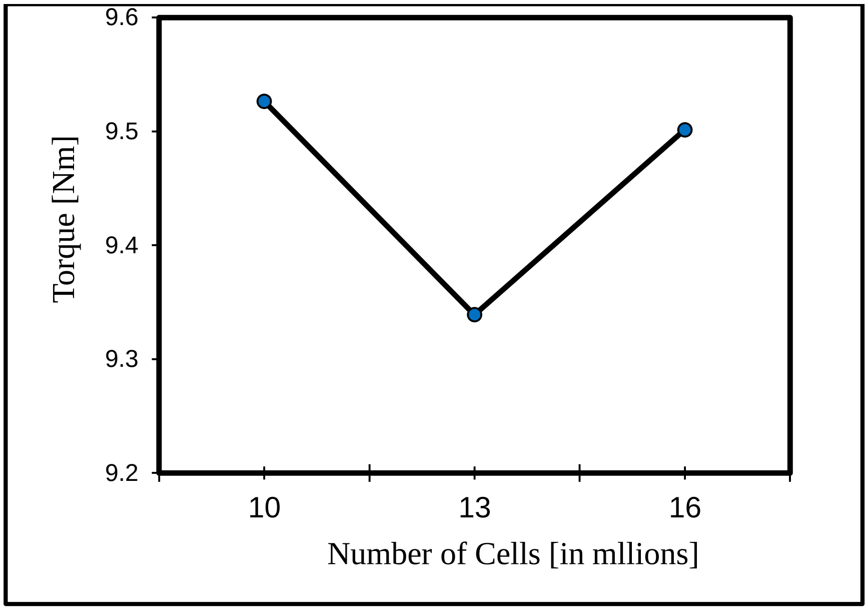


Figure 16: Grid Independence Test

Since, following the grid independence test, increasing the grid intensity, had almost little effect on the torque, but since this is a small wind turbine simulation, a small change in torque also, had a great effect on Cp calculation, due to the percentage difference, therefore CFD results for a small wind turbine to find Cp require a lot refiner grid and high computational resources.

3.7 References:

[1] M. M. et al., aerodynamics experiment phase VI: wind tunnel test configurations and available data campaigns: NREL Golden, USA, 2001

[2] Abolfazl Pourrajabian et al, Applying micro scales of horizontal axis wind turbines for operation in low wind speed regions, Energy Conversion and Management, 2014

[3] Fluent 19.1 User’s Guide

[4] M. O. L. Hansen et al., "State of the art in wind turbine aerodynamics and aero elasticity," Progress in aerospace sciences, 42 vol., 2006.

[5] W. Z. Wang Jian et al., "Determination of fire scenario in performance-based fire safety design," FST, 2005.

[6] Yao Wei, Liao Guangxuan, “Grid-independent Issue in Numerical Heat Transfer” State Key Lab. of Fire Science, University of Science and Technology of China

Chapter # 4: Results and Discussion

4.1 Validation

Commercially available CFX code is used to solve the numerical equations. To validate the numerical model, the CFD results are then compared with the BEMT results [1].

The main objective of this thesis is to validate the BEMT designed rotor, and to validate and compare the power produced and more importantly, Coefficient of performance, and to find out whether BEMT is a reliable tool to design Wind turbine rotors. According to Abolfazl et al., the wind rotor designed at tip speed ratio of 5.7 achieves a maximum of 0.481 Cp, whereas the CFD calculation run at designed conditions achieve a CP of 0.47, which is less than the Betz limit of 0.593.

The following figure 17 shows the relationship between Tip speed ratio and Cp. It is a good practice to analyze the CFD results of power, torque etc. in terms of non-dimensional, tip speed ratio.

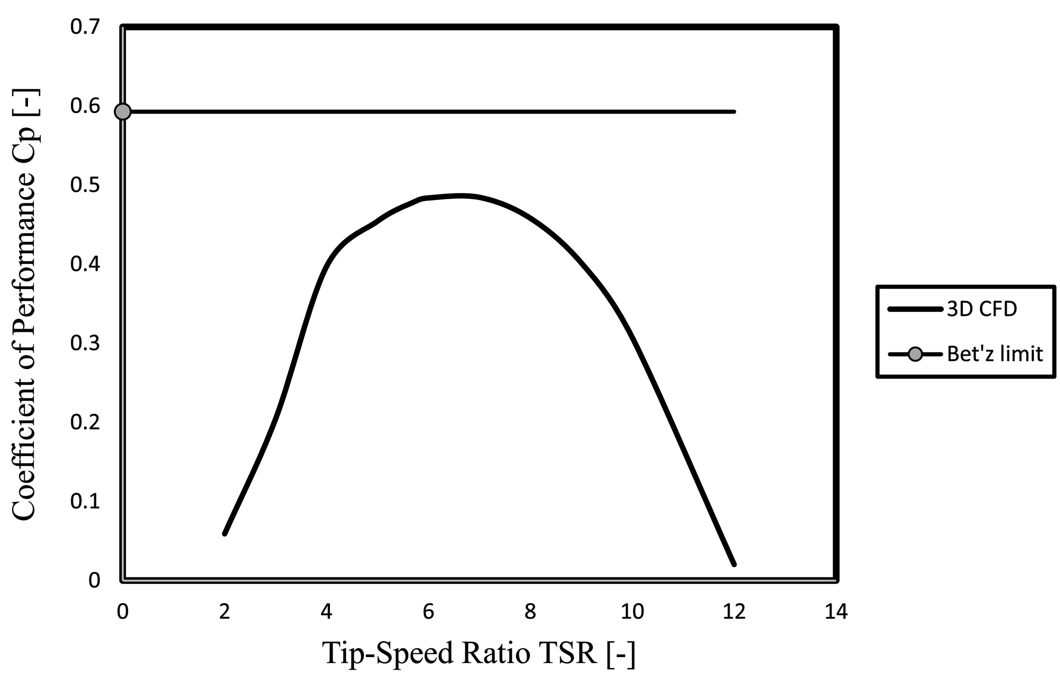


Figure 17: Cp vs TSR Curve

According to Wood [2], small wind turbines could practically achieve a Cp of 0.48 in the physical environment, therefore the BEMT algorithm was designed to find a design which was able to achieve a Cp of 0.48, whereas CFD results shows that a higher designed TSR could achieve a higher Cp but it doesn't guarantee whether it would materialize practically.

The effect of wind speed is also analyzed by changing the input velocity across the domain. Following table shows the effect of wind speed on Power and torque produced.

Wind Speed [m/s]	***Torque [Nm]***	***Power [kW]***
1	0.06	0.008
2	0.3225	0.009
4	1.45	0.082
6	3.36	0.285
8	6.05	0.685
9	7.67	0.978
10	9.52	1.348
12	13.79	2.343
13	16.23	3.751
14	18.92	5.628

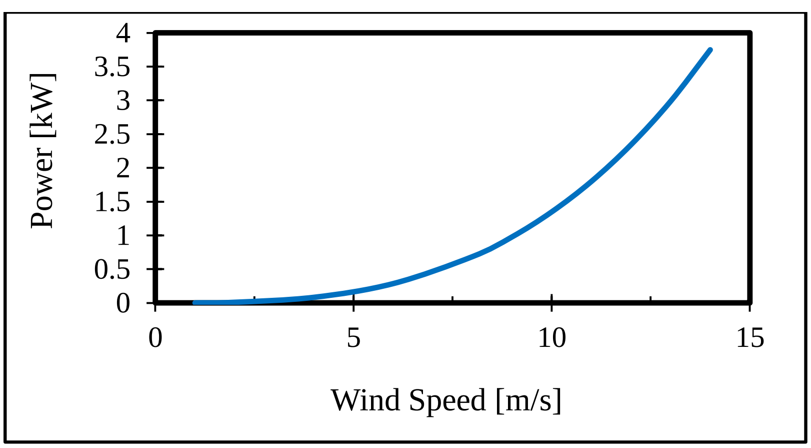


Figure 18: Power vs. Wind speed curve

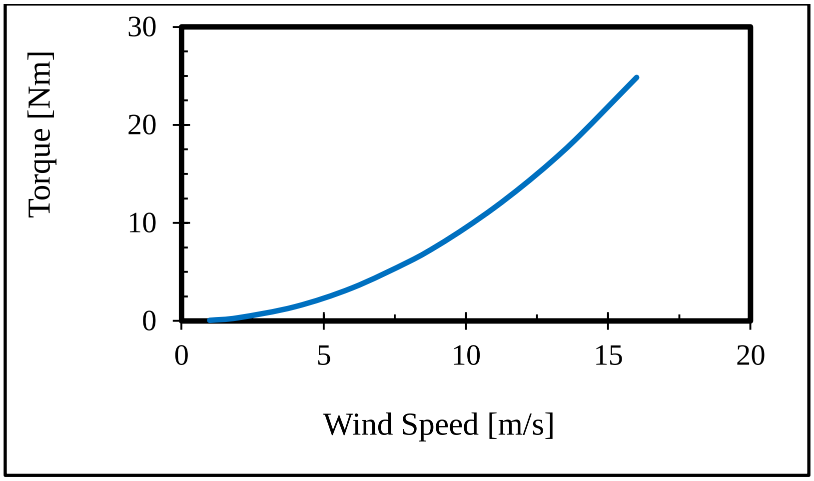


Figure 19: Torque vs. Wind speed curve

Figure 18 & 19 shows the relationship between torque and power with changing wind speed. It illustrates that torque and consequently the power increased swiftly as the wind speed increased.

4.2 Post-CFD Results

The numerical results of the simulation are processed through Post-CFD feature of ANSYS and TECPLOT 360. The post processed results consists of pressure and velocity contours on suction and pressure side of the blade across different span ratio across the blade at different tip speed ratio, induced velocities up and downstream of the rotor blade, and streamlines across the blade at different tip speed ratio.

4.2.1 Induced velocity

Figure 20 and 21 shows the velocity along the wind direction at different Tip Speed ratio, with a plane cutting the turbine blade at an angle of 60^{o}. It can be deduced at design tip speed ratio of 5.7, maximum energy is extracted from wind, whereas at TSR 3 most of the energy i.e. wind velocity remains intact. To use this turbine as in a wind farm, more research needs to be done.

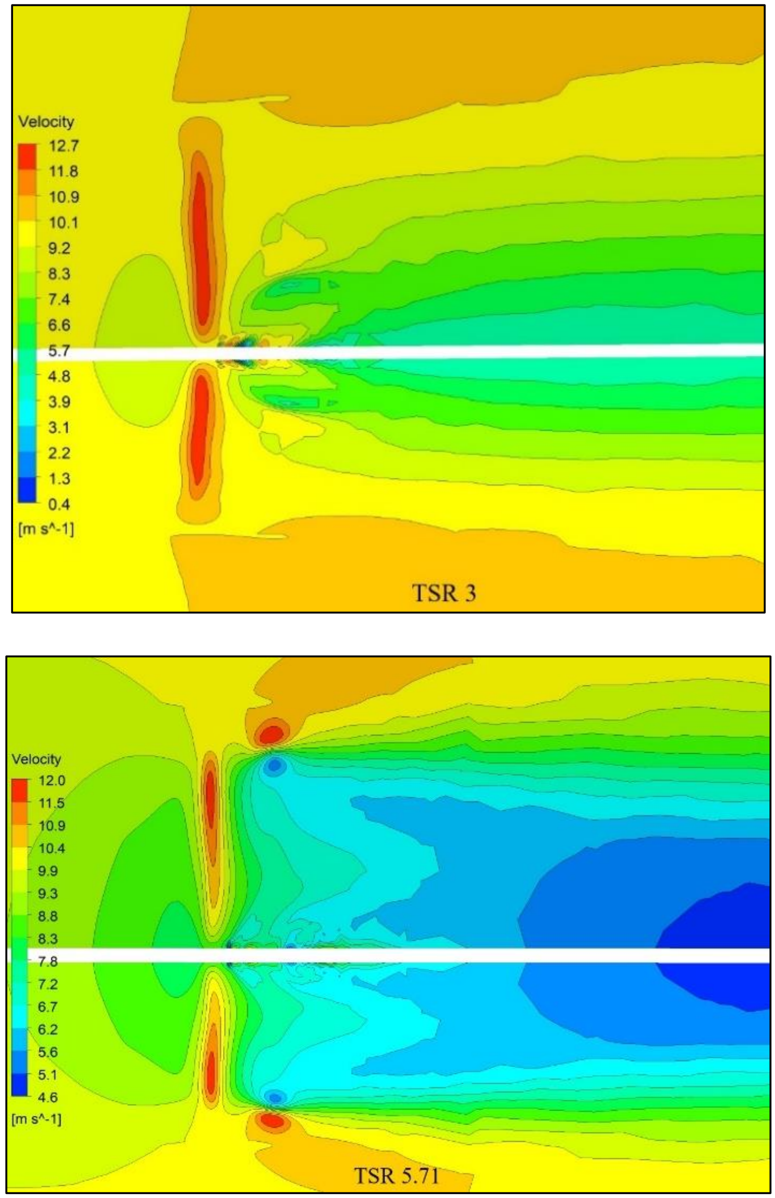


Figure 20: Induced axial Velocity at TSR 3 & 5.71

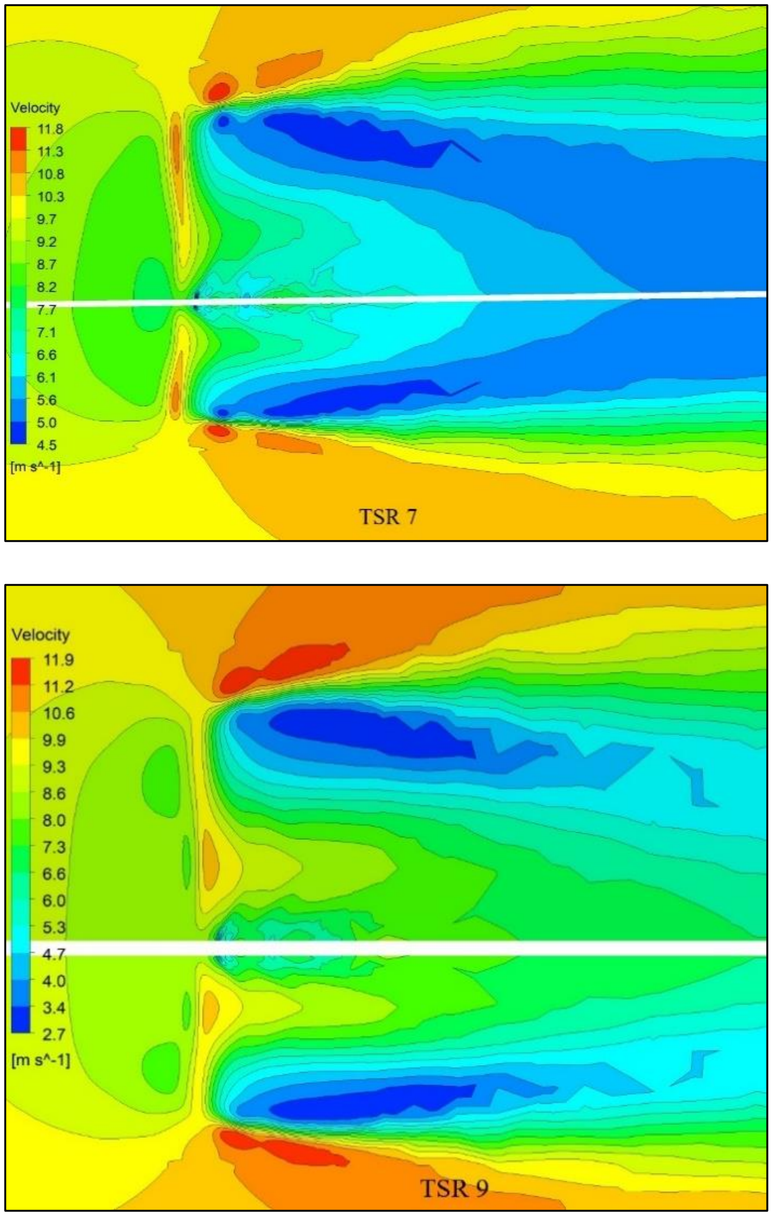


Figure 21: Induced axial Velocity at TSR 7 & 9

4.2.2 Pressure and Velocity Contours

Figure 22-25 shows pressure and velocity contours at different span ratio across the blade for TSR 5.71, 7, 9 and 3, respectively.

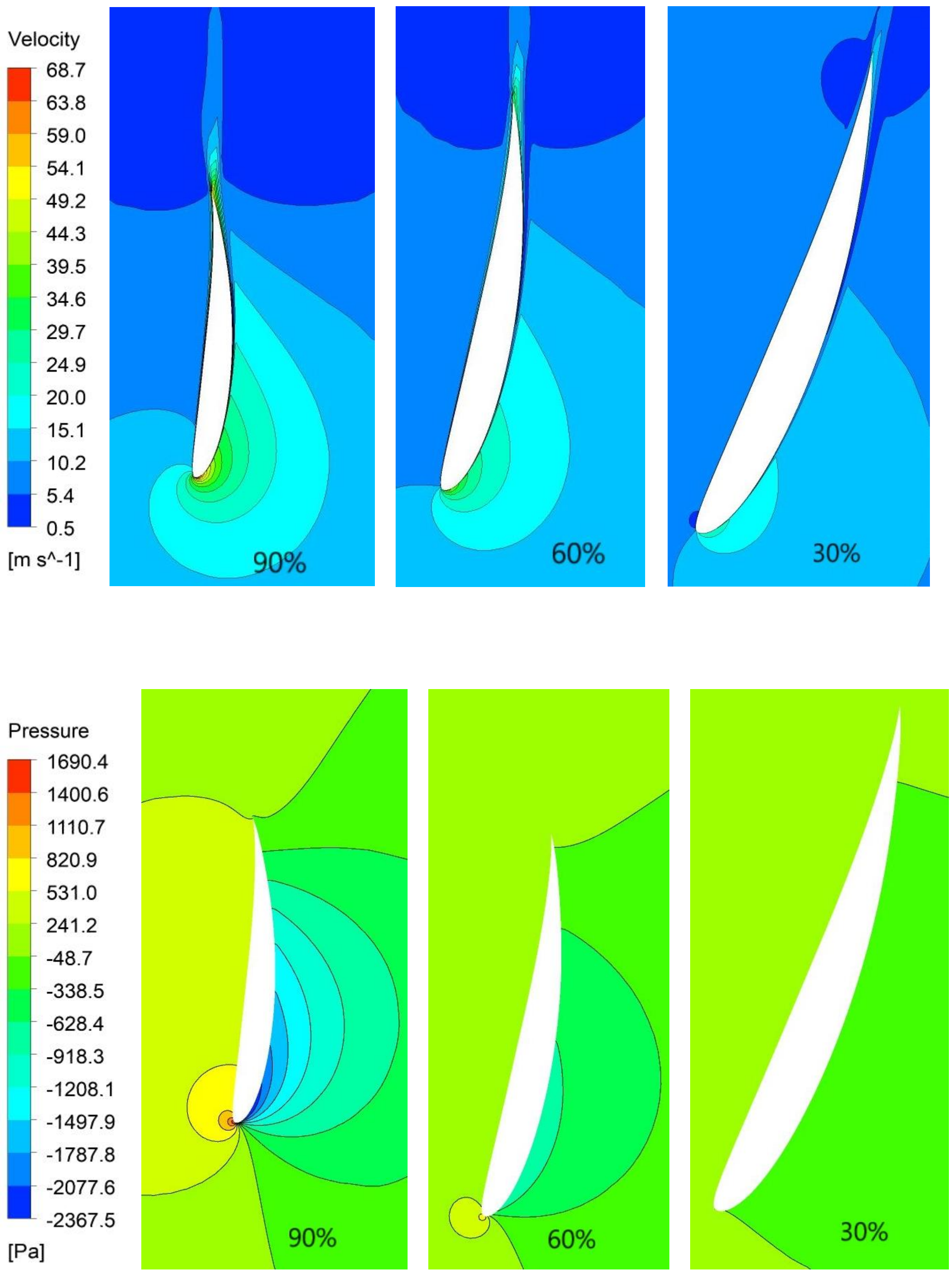


Figure 22: Velocity and Pressure Contours at TSR 5.71

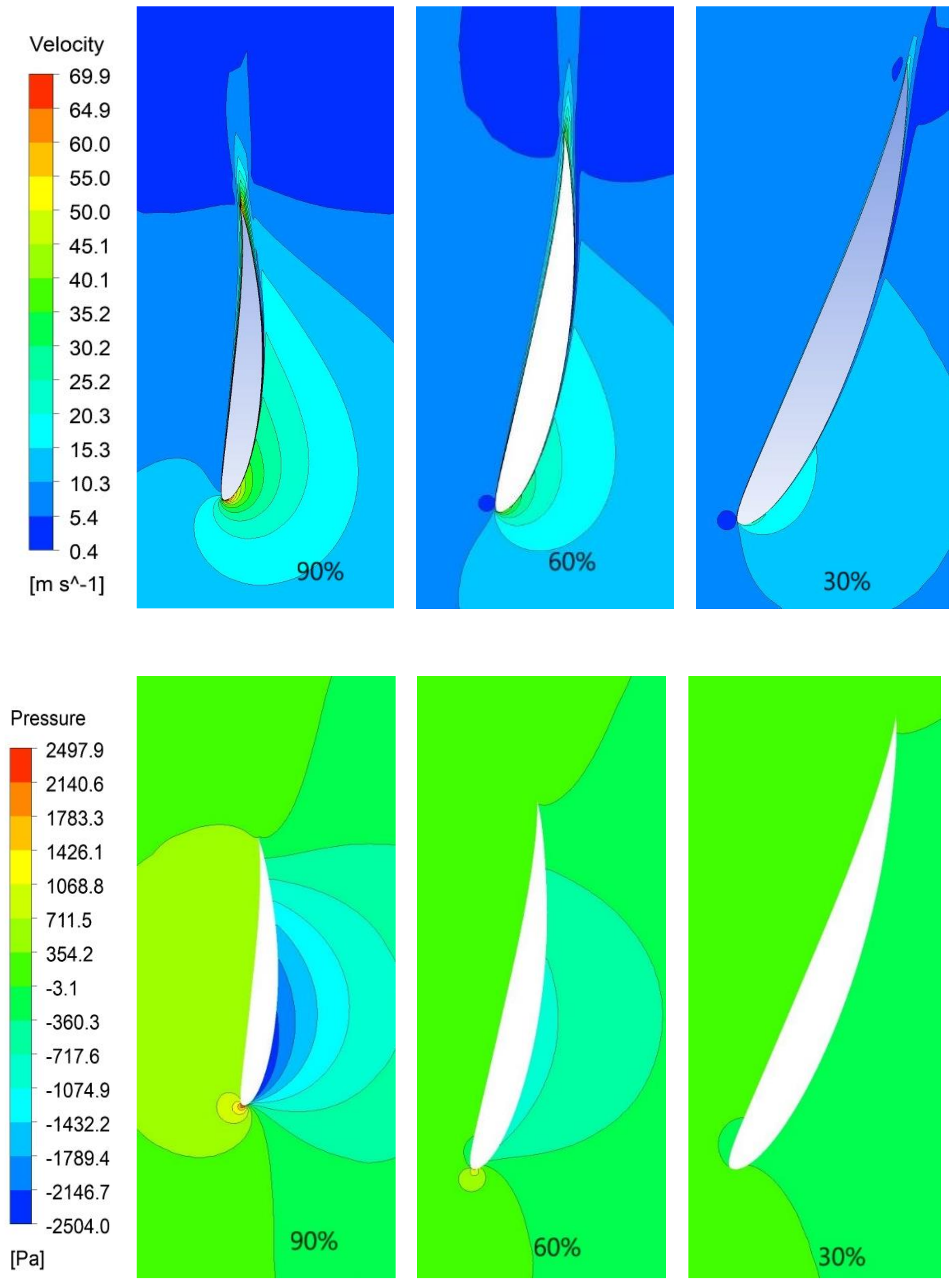


Figure 23: Velocity and Pressure Contours at TSR 7

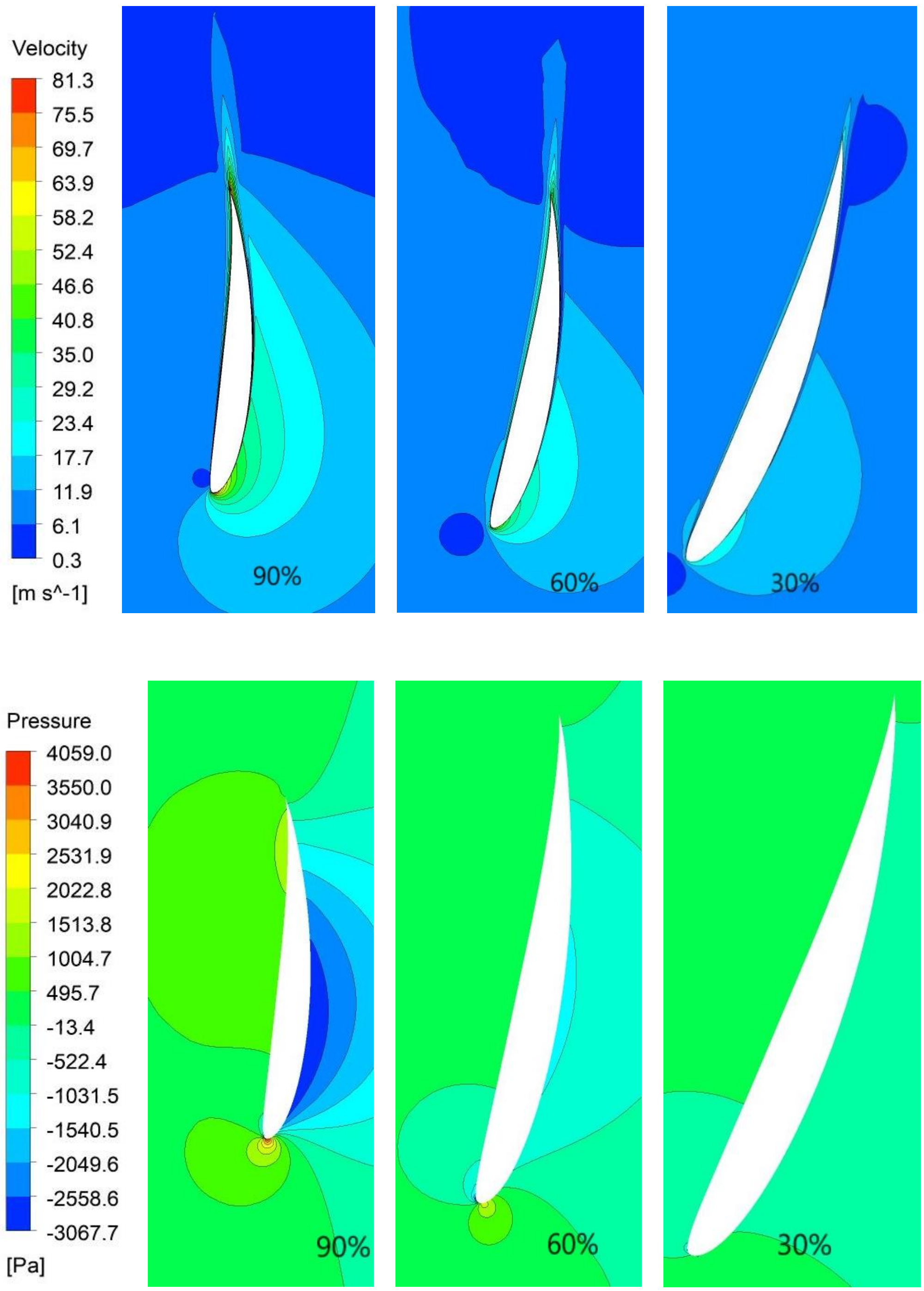


Figure 24: Velocity and Pressure Contours at TSR 9

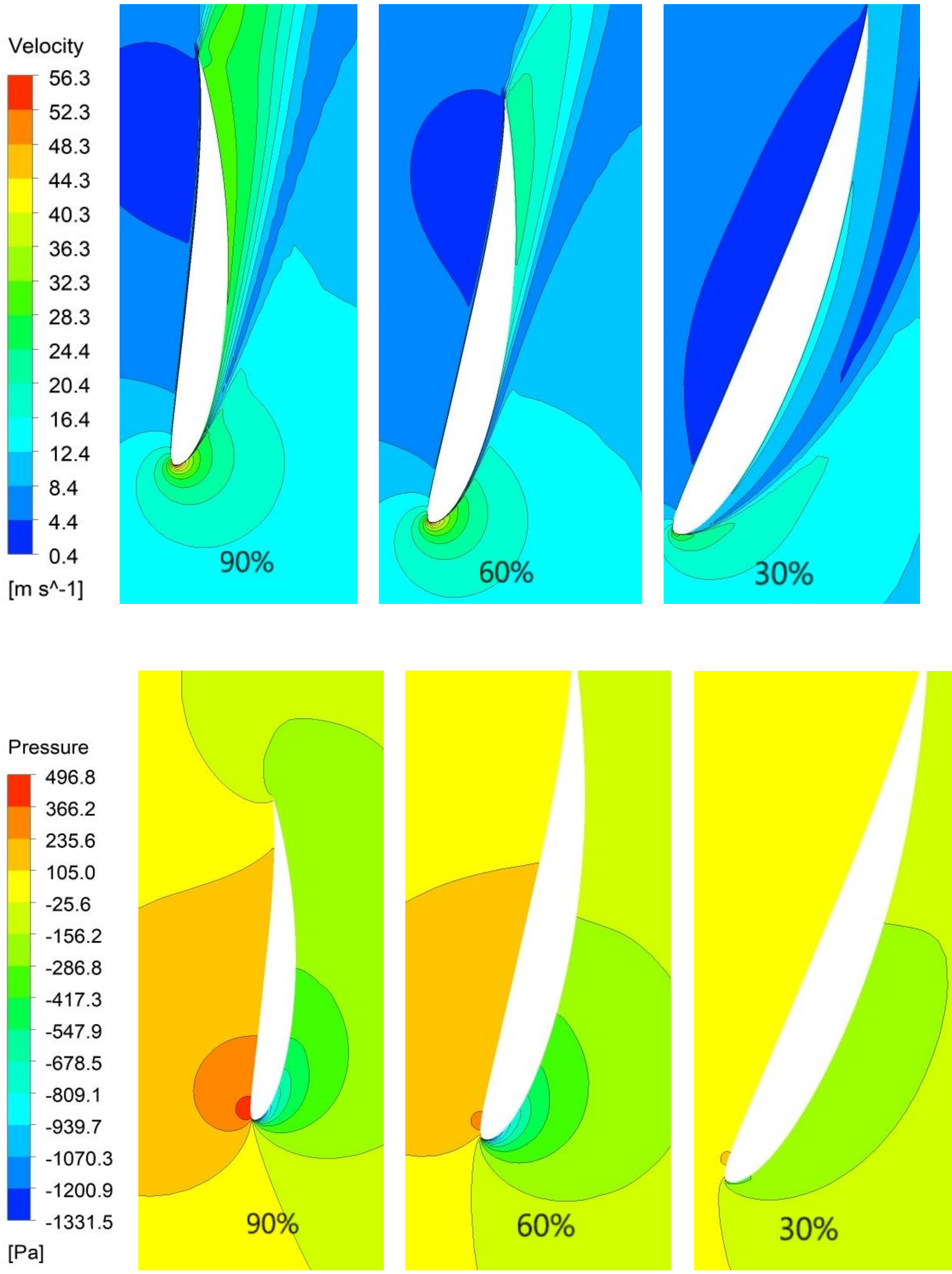
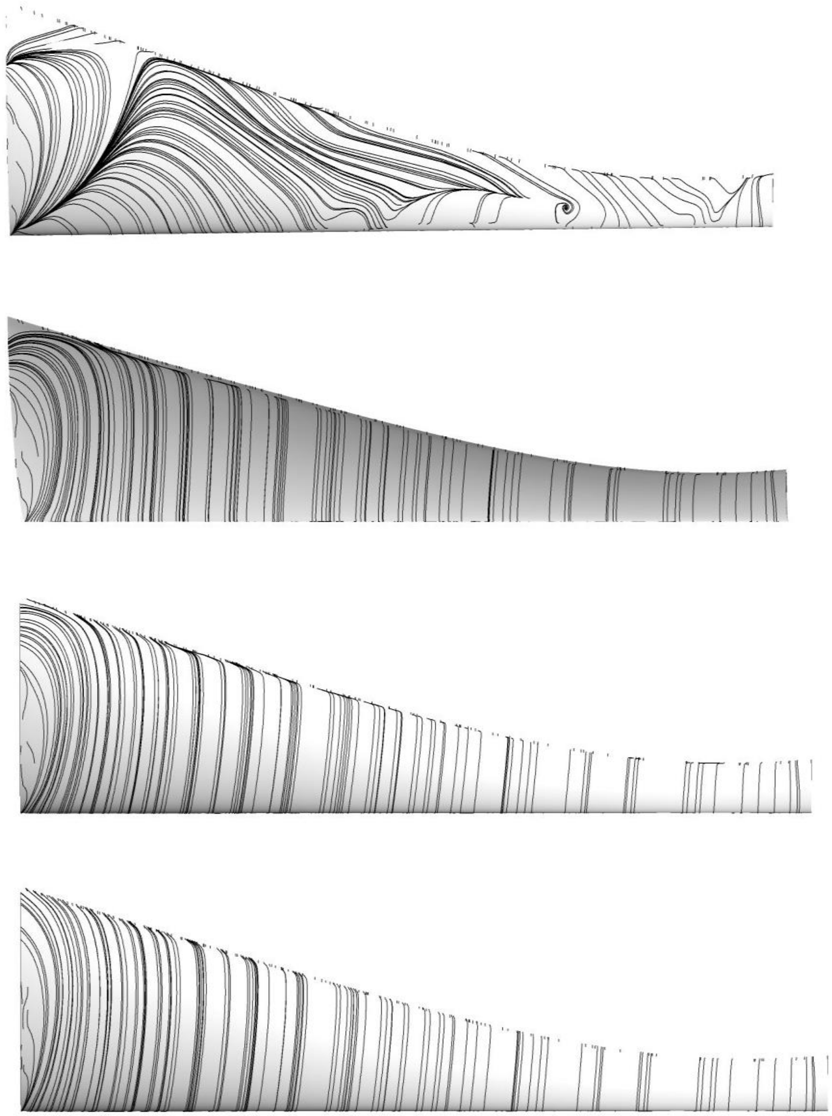


Figure 25: Velocity and Pressure Contours at TSR 3

4.2.3 Surface Streamlines



TSR 3

TSR

5.71

TSR 7

TSR 9

Figure 26: Streamlines on Suction side of blade

4.3 Conclusion

This research focused on the CFD validation of small HAWT design for operation in speed regions with low wind speed. The difference between the Cp from BEMT and CFD were justifiable considering the tip losses and 3D rotational losses compared to Blade element momentum theory. At rated wind speed of $10ms^{-1}$ the power coefficient is evaluated to be 0.47, which concurs from the literature and the BEMT result. Secondly, at starting wind speed of the rotor, the rotor produces power of about 285 W, which is considerable considering the size of turbine. The SG 6043 airfoil based small wind turbine at its rated wind speed is able to produce 1 kW, which can help an individual home to achieve energy security. In Pakistan, these wind turbines will be feasible in some areas of Baluchistan and Sindh, where wind speeds can go up to 10ms-1.

4.4 References

[1] Abolfazl Pourrajabian et al, Applying micro scales of horizontal axis wind turbines for operation in low wind speed regions, Energy Conversion and Management, 2014

[2] Wood DH, Small wind turbines: analysis, design, and application, green energy and technology. 2011.

List of Publications

1. Jawad A., Adeel J. (2018) "CFD Evaluation of 1kW Small-Scale Horizontal Axis Wind Turbine for Operations in Low Wind Speed Regions" 17th World Wind Energy Conference, Karachi, The World Wind Energy Association
2. Jawad A., Adeel J. (2018) "CFD Evaluation of 1kW Small-Scale Horizontal Axis Wind Turbine for Operations in Low Wind Speed Regions" High Performance Energy Efficient Buildings and Homes HPEEBH 2018 August 1-2, 2018 Lahore, Pakistan