

CFD INVESTIGATION OF THE EFFECTS OF SPIKE AND DINO-SHELL VORTEX GENERATORS ON THE AERODYNAMIC PERFORMANCE OF A WIND TURBINE BLADE

Submitted in partial fulfillment of the requirements for the award of

Bachelor of Engineering Degree

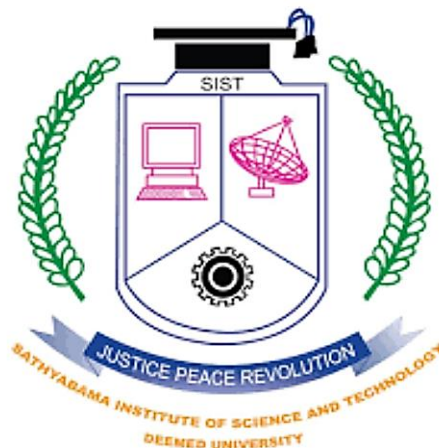
In

Aeronautical Engineering

By

NEERAJA MEDIM (Reg. No. 38260028)

K.NEVIS AMMAL RAJ (Reg. No. 38260029)



DEPARTMENT OF AERONAUTICAL ENGINEERING

SCHOOL OF MECHANICAL ENGINEERING

SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

Accredited with grade “A” by NAAC

JEPPIAAR NAGAR, RAJIV GANDHI SALAI, CHENNAI – 600119

MAY 2022



SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited with "A" grade by NAAC
Jeppiaar Nagar, Rajiv Gandhi Salai, Chennai - 600 119
www.sathyabama.ac.in



DEPARTMENT OF AERONAUTICAL ENGINEERING

BONAFIDE CERTIFICATE

This is to certify that this Project Report is the bonafide work of NEERAJA MEDIM (Reg. No. 38260028) and K.NEVIS AMMAL RAJ (Reg. No. 38260029) who carried out the project entitled **"CFD INVESTIGATION OF THE EFFECTS OF SPIKE AND DINO-SHELL VORTEX GENERATORS ON THE AERODYNAMIC PERFORMANCE OF A WIND TURBINE BLADE"** under our supervision from September 2021 to May 2022.

Internal Guide

Dr.S.MANIGANDAN, M.E., PhD.

Head of the Department

Dr. P. BOOMADEVI, ME., PhD.

Submitted for Viva voce Examination held on 05.05.2022

Internal Examiner

External Examiner

DECLARATION

We, NEERAJA MEDIM (Reg.No.38260028) and K.NEVIS AMMAL RAJ (Reg.No.38260029) hereby declare that the Project Report entitled “**CFD INVESTIGATION OF THE EFFECTS OF SPIKE AND DINO-SHELL VORTEX GENERATORS ON THE AERODYNAMIC PERFORMANCE OF A WIND TURBINE BLADE**” done by us under the guidance of **Dr.S.MANIGANDAN, M.E., PhD**, is submitted in partial fulfillment of the requirements for the award of Bachelor of Engineering Degree in Aeronautical Engineering.

DATE: 05.05.2022

PLACE: CHENNAI

1. NEERAJA MEDIM



Neeraja Medim

2. K.NEVIS AMMAL RAJ



SIGNATURE OF THE CANDIDATE

ACKNOWLEDGEMENT

We are pleased to acknowledge our sincere thanks to the Board of Management of **SATHYABAMA** for their kind encouragement in doing this project and in completing it successfully. We are grateful to them.

We convey our sincere thanks to **Dr.S.Prakash, M.E., Ph.D.**, Dean, School of Mechanical Engineering and **Dr.P.Boomadevi M.E., Ph.D.**, Head of the Department, Department of Aeronautical Engineering for providing us necessary support and details at the right time during the progressive reviews.

We would like to express our sincere and deep sense of gratitude to our Project Guide **Dr.S.MANIGANDAN, M.E., PhD**, for his valuable guidance, suggestions and constant encouragement paved way for the successful completion of our project work.

We also express our thanks to all the Teaching and Non-teaching staff members of the Department of Aeronautical Engineering who were helpful in many ways for the completion of the project.

ABSTRACT

In the era of modernizing available resources to extract the best from it, attempts are being made to revolutionize the world's dependence on renewable sources of energy and avoid carbon emissions from electricity usage as much as possible. To make certain safety of structural standards underneath intense situations, regardless of the ensuing rate of energy loss, airfoils are thicker at the root followed by tapered profiles to the tip. To avoid this loss, a passive flow manipulative tool referred to as a vortex generator (VG) is mounted. For large reduction in the power generating performance of a wind turbine is due to the surface roughness, which can be avoided to some part, by using Vortex generator arrays, in between 14%-41% of the blade span that increases AEP by 1%, on the beginning point of stall to govern the flow subject close to the wall of the suction surface. Computational Fluid Dynamics has been used here to research the aerodynamic traits induced as an end result of the form and format of the VG on a multi-MW wind turbine blade. The separated and vortical flow conduct at the suction floor of the wind turbine blade of length 60 m with both spike and dinoshell type VGs was watched by SST K-epsilon turbulence model. The study about how parameters of VG shape affect, and based on these effects, the RSM was used to research the effect on the layout parameters of the VG. Primarily, based on the CFD outcomes, the VG design parameters have been selected by thinking about the lift and vorticity above the trailing aspect. It is advised that VGs can be used to enhance wind turbine overall performance and result is 1% growth in AEP by ways of suppressing the drift separation and mitigating the soiling effect. The location of vortex generators along the blade span is essential. The outcomes of chord wise role, span wise spacing and VG length were studied with force balance measurements of a 2D wing phase, reducing the distance among adjacent VGs producing drastic increase in the static stall angle and L_{max} , also large increase in drag. The most suitable chord wise position of the vortex turbines discovered to be in the range of 15-20% of x/c , where a comparatively low parasitic drag and a smooth put up stall lift curve had been carried out. The variation in velocity, pressure and turbulence pathlines have been observed by evaluating contour figures and plots, using ANSYS FLUENT Software.

TABLE OF CONTENTS

CHAPTER No.	TITLE	PAGE No.
	ABSTRACT	V
	LIST OF FIGURES	VIII
	LIST OF TABLES	X
	LIST OF SYMBOLS AND ABBREVIATIONS	XI
1	INTRODUCTION	12
	1.1 BACKGROUND	13
	1.2 BENEFITS	14
2	LITERATURE REVIEW	16
3	AIM AND SCOPE	19
	3.1 AIM AND SCOPE	19
4	METHODOLOGY	20
	4.1 GRID DEVELOPMENT	20
	4.2 NUMERICAL METHODOLOGY	20
	4.2.1 TURBULENCE MODELS	22
	4.2.2 CHOOSING A TURBULENCE MODEL	22
	4.2.3 STANDARD k- ϵ MODEL	22

CHAPTER No.	TITLE	PAGE No.
	4.2.4 BOUNDARY PLANE	23
	4.2.5 Solver and Operating Condition Defining	26
	4.2.6 BOUNDARY CONDITIONS	27
	4.2.7 REPORT REFERENCE VALUES	27
5	DESIGN AND CFD ANALYSIS	28
	5.1 SINGLE-SIDED SPIKE VORTEX GENERATOR BLADE	30
	5.2 SINGLE-SIDED DINOSHELL VORTEX GENERATOR BLADE	38
	5.3 NORMAL HAWT BLADE	40
6	RESULTS AND DISCUSSION	43
7	SUMMARY AND CONCLUSION	49
8	REFERNCES	50

FIGURE No.	TITLE	PAGE No.
1.1	Effect of vortex generators	15
2.1	Aerodynamic impact on airfoil lifts due to LE roughness	18
4.1	Velocity Inlet	24
4.2	Pressure Outlet	24
4.3	Meshing	25
5.1	NACA 0015 Airfoil	28
5.2	Array of Spike VGs on a HAWT Blade	30
5.3	Thickness = 250mm Dimension of Spike vortex generator	30
5.4	Horizontal axis wind turbine blade with Spike vortex generators designed in CATIA	31
5.5	Scaled Residuals graph plotted for 500 iterations using ANSYS for Spike VGs Installed HAWT Blade	31
5.6	Velocity Pathlines for single-sided spike VGs installed HAWT Blade	32
5.7	Pressure Pathlines for single-sided spike VGs installed HAWT Blade	32
5.8	Turbulence Pathlines for single-sided spike VGs installed HAWT Blade	33
5.9	Computational Report of Aerodynamic Parameters of Spike VGs HAWT Blade	33
5.10	Comparison of GH Bladed Design Data and CFD Results	34

FIGURE No.	TITLE	PAGE No.
5.11	Cross-sectional velocity contour of the blade: (a) without VG; (b) with VG	35
5.12	(a)Lift Coefficient vs. Angle of Attack, (b) Drag Coefficient vs. AoA	35-36
5.13	Vorticity contours at the (a) midpoint of the chord and (b) Trailing edge	38
5.14	HAWT Blade with Dinoshell vortex generators	38
5.15	Scaled Residuals graph plotted for 500 iterations using ANSYS for Dinoshell VGs Installed HAWT Blade	39
5.16	Velocity Pathlines for single-sided dinoshell VGs installed HAWT Blade	39
5.17	Pressure Pathlines for single-sided dinoshell VGs installed HAWT Blade	40
5.18	Normal HAWT Blade of length 60 m	41
5.19	Scaled Residuals Graph Plotted for 200 iterations using ANSYS for normal HAWT Blade	42
5.20	Velocity Pathlines for normal HAWT Blade	42
6.1	Power Curves of a typical 5MW wind turbine	44
6.2	Flow characteristics near VG and vortex formed	44
6.3	Aerodynamic effects on blade (a) without VGs (b) with VGs	45
6.4	Fabrication and Experimentation	46

LIST OF TABLES

Table No.	Name	Page No.
1.1	Classification of wind turbines	12
4.1	Solver Specifications	26
4.2	Material Properties	26
4.3	Velocity Inlet and Pressure Outlet Conditions	27
4.4	Report Reference Values	27
5.1	Wind Turbine Specifications	29
5.2	Specifications of VG with respect to chord length	29
5.3	BEMT Design Data and CFD Results	34
5.4	Case Studies Results to select VG Design Parameters	36
6.1	Experimental Readings	47
6.2	C_p Port Readings	47
6.3	Results of C_l , C_d at 100RPM, 200RPM and 300RPM	48

LIST OF SYMBOLS AND ABBREVIATIONS

CATIA Computer Aided Three-Dimensional Interactive Application

ANSYS Analysis of Systems

SCADA Supervisory Control and Data Acquisition

b Wing Span[m]

c Wing chord[m]

C_L Lift coefficient

C_D Drag coefficient

D Drag

L Lift

α Angle of Attack, AoA

ρ Density of air

C_p Betz coefficient

V Velocity, [m/s]

VGs Vortex Generators

AEP Annual Energy Production

LE Leading Edge

TE Trailing Edge

CAD Computer Aided Design

RSM Response Surface Methodology

CHAPTER 1

INTRODUCTION

The devices that convert the kinetic strength of winds blowing over the earth's surface into electrical energy that power our households and farms to industries are called wind turbines. Wind farms (masses of hundreds of large generators, in installations) now generate over 650 Giga Watts of energy, almost 60 GW produced every year. It is believed that the more wind power we use, the more geopolitical competition over vital materials for windmills for example neodymium will increase. However, this mindset has been criticized for failing to realize that maximum wind turbines do not use permanent magnets and for underestimating the electricity of monetary incentives for multiplied production of these minerals. There may be more and more crucial source of intermittent renewable sources that are utilized in many countries to decrease expensive energy demands and decrease reliance on fossil fuels. One examine claimed that, as of 2009, wind had the "least relative greenhouse gasoline releases, the least water consumption needs and...the most favorable social affects" compared to photovoltaic, hydro, geothermal, coal and gas energy sources. Smaller wind generators are used for programs inclusive of battery charging for auxiliary strength for boats or caravans, and to electricity visitors caution symptoms. Larger turbines can make a contribution to the domestic energy supply even as selling unused energy back to the utility dealer through the electric grid. Wind energy converters come in a variety of sizes, with either horizontal or vertical axes. Wind Power Density is a term that quantifies the amount of wind power that can be generated over any area. It is the suggested annual energy available per unit area swept by the turbine. Calculation of wind energy density involves the effects of wind speed, v and air density, ρ . Windmills are categorized by the wind speed they are designed for and with class A to C regarding the turbulence depth of the wind.

Class	Average Wind Speed (m/s)	Turbulence Intensity
IA	10	16%
IB	10	14%
IC	10	12%

Table 1.1 Classification of wind turbines

Our study involves the above classification of wind turbines.

From law of conservation of mass, $m_{in}=m_{out}$ in a turbine. And according to Betz's law, the maximum wind power extracted by a wind turbine is $\frac{16}{27}$ (59.3%) of the kinetic energy rate of the air that is arriving at the turbine. Therefore, the maximum theoretical power output, P of the turbine is $\frac{16}{27}$ times the rate at which kinetic energy of the air arriving with velocity, v and density, ρ at its effective disk area, A , that is

$$P = \frac{16}{27} \frac{1}{2} \rho v^3 A = \frac{8}{27} \rho v^3 A,$$

The factors affecting the ultimate costs of wind power are the wind-to-rotor efficiency (along with the friction and drag associated with the rotor blade). In addition to these inefficiencies such as, generator and converter losses, gearbox losses, downsize the total power generated. To protect components from undue put on, extracted strength is kept constant above the rated operating speed as theoretical power increases at the rate of wind speed raised to the power of three, similarly lowering theoretical performance. Efficiency can decrease slightly through the years, one of the major reasons being dust and insect carcasses on the blades which alters the aerodynamic profile and essentially reduces the raise to drag ratio of the airfoil.

1.1. BACKGROUND

Intrigued in enhancing the overall performance of wind generators with the aid of optimizing the techniques used for blade design and flow controlling devices has grown in the previous couple of decades. In specific, several studies have centered on refining those methods to increase the efficiency of wind generators. Wind turbine manufacturers have attempted to enlarge the blade length to improve the aerodynamic performance, however additionally to decrease electricity consumption.

In many of the advanced technologies, blade aerodynamics is of incredible importance, as the wind electricity is produced by the aerodynamic torque from the blades. Blade design is a compromise of aerodynamics and systems, with which the aid of rule-of-thumb, is ready by using systems dominating the inboard blade up to 30% span and aerodynamics ruling at the outer 50% span. The inboard segment of the blade is designed for structures and much less attention is contributed to aerodynamics.

Historically, the wind enterprise has carried out aerodynamic blade design and performance analysis based on Blade Element Momentum principle (BEMT). Aerodynamic blade design techniques for wind generators were reviewed previously. The layout method based totally on BEMT applies numerous forms of loss-creation fashions to the end and hub of the blade, in which the 3D glide field is dominant. However, the problem of aerodynamic reliability of the blade with excessive accuracy persists, as a result of the expansion of its length and the taper of its tip.

1.2. BENEFITS

The utility of an array of vortex generators (VG) to control glide separation and mitigate blade soiling issues of wind mills changed into studying the usage of the industrial CFD software program. In this effort, VGs are used as a passive overall performance-improving device for commercial purpose wind turbine is studied. A VG is a passive flow manipulative tool in the shape of a small vane and is in particular used to delay the stall occurrence on an airplane wing or by creating longitudinal vortices to enhance heat transfer. VGs are installed in the direction of root and suction floor of the blade of a wind turbine to mitigate airflow separation. Many CFD analyses and wind tunnel assessments have been done to evaluate the feasibility of VG adoption with wind turbine blades, but few CFD research centered on full-scale wind generators equipped with Gurney flap VGs other than airfoil based simulation and wind tunnel testing. The aerodynamic traits of multi-MW horizontal-axis wind turbine blades have been analyzed by using the industrial CFD software ANSYS FLUENT. The outcomes of the analysis were confirmed with experimental data. These researches have optimized the form and format of the VG to maximize the overall wind turbine performance in terms of how much power it is generating.

Three major numerical analysis strategies, namely, direct numerical simulation (DNS), big eddy simulation (LES), and RANS simulation, may be used to analyze turbulent flow fields. To investigate the overall vortex behavior precisely in a turbulent flow field discipline containing vortices of diverse scales, the calculation grid size have to be smaller than the minimal space scale of the vortex structure and the time taken should be less than the minimal time scale of the vortex variation. LES uses the spatially averaged Navier-Stokes equation which immediately calculate vortices extra than the grid scale, not implying sub grid-scale version for vortices smaller than the grid scale. DNS and LES require huge

resources for their calculations, making them unfeasible in realistic engineering applications. In contrast, RANS yields a decrease decision evaluation than DNS or LES; it's far extensively used in engineering applications as it does no longer require excessive-resolution calculation grids. The consequences of chord wise function, span sensible spacing and VG size have been studied with pressure stability measurements of a 2D wing phase. Lowering the gap among adjoining VGs produced huge increase in stall angle, and L_{max} , however additionally led to a big growth in drag in addition to sharp lift surges at angles exceeding the static stall perspective. The most efficient chord wise role of the vortex mills became determined to be in the range of $x/c = 15\% - 20\%$, where a relatively low parasitic drag and a smooth post-stall carry curve were completed. Smoke flow visualization techniques had been conducted at numerous chord wise positions to provide insight into the interaction between adjacent stream wise vortices. The experimental aerodynamic overall performance curves of the choicest VG configuration had been used to project their impact on wind turbine blade aerodynamics. Three one-of-a-kind rotor blades were designed and numerous stall and pitch regulated wind turbine models were simulated via a Blade detail Momentum (BEM) code (Q Blade) developed by way of smart Blade GmbH. The performances of the rotor blades with and without VGs become simulated with a view to check their effects at different loads. K- ϵ standard wall function turbulence model has been used and validated with the commercial computational fluid dynamics (CFD).

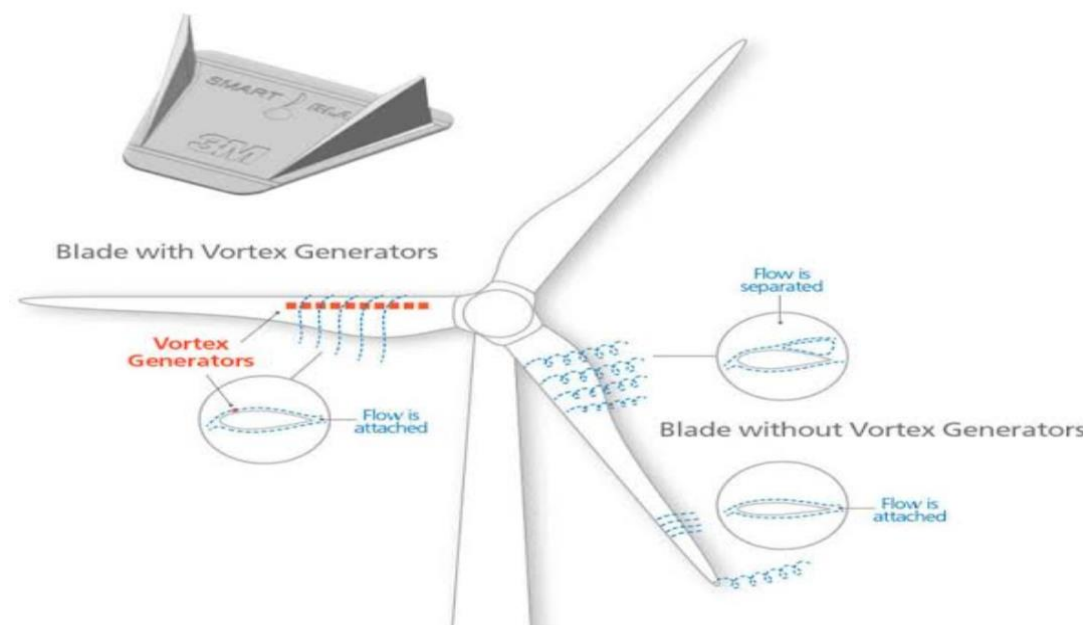


Fig.1.1.Effect of vortex generators on the fluid flow over a blade

CHAPTER 2

LITERATURE REVIEW

1. Experimental Study of Airfoil Performance with Vortex Generators by M. B. Bragg and G. M. Gregorekt

- Vortex generators were used to retrieve the lost lift of an airfoil due to the early transition and roughness from the leading edge roughness simulator. Their purpose is to introduce vortices into the flow ahead of the separation point to energize the boundary layer and thus to prevent the flow separation.
- The airfoil was designed for a canard cruise Cl of 0.9. At this condition, the vortex generators return the lift within approximately 0.1 of the natural transition level, depending on the type of vortex generators used. A significant loss in $C_{l, \max}$ is seen even with the presence of vortex generators, although significant lift at these high angles has been restored so as not to affect seriously the aircraft operation.
- Three types of vortex generators were used: SMALL VORTEX GENERATOR, LARGE VORTEX GENERATOR, and DELTA VORTEX GENERATOR.
- For the tripped airfoil, all four vortex generator configurations increased the lift and reduced the drag. The delta plan form vortex generators caused only a very small increase in drag on the airfoil with natural transition. Since on the tripped airfoil these vortex generators also restored the necessary lift.

2. CFD-Based In-Depth Investigation of the Effects of the Shape and Layout of a Vortex Generator on the Aerodynamic Performance of a Multi-MW Wind Turbine by Hyeon-Gi Moon, Sunho Park, Kwangtae Ha and Jae-Ho Jeong

- There was a study using CFD analysis of the VG-mounted wind turbine blade by RANS, SST k- ω equation, blade length of min.35 m, and a cuboid VG was inspected for the lenience of lattice formation. Although, some studies show that calculated power increases via BEM using the lift and drag coefficient, BEM calculations are less precise than those of RANS CFD. Analysis of the aerodynamic

characteristics of multi-MW horizontal-axis wind turbine blades is done using the commercial CFD software ANSYS CFX. These results were validated with GH-Bladed design data. These researches optimized the shape and layout of the VG to maximize the wind turbine performance in terms of how much power is generated.

- The distance between the VG pairs (λ) and the angle of attack (α) of the VG were specified as 6° and 11° for efficiency. The recommended λ/h investigated through a flat-plane CFD analysis study is 5.
- The VGs proved to play an important role by improving the aerodynamics of multi-MW wind turbines by 2.80% at the rated wind speed. This was confirmed by the simulation at the rated wind speed with more than 55 million elements. 5. The results of this study are useful for selecting the shape and mounting position of the VG by performing a full-scale simulation by automating the construction of the simulation environment. The selected VG discussed in this study is currently installed in a 2.3 MW wind turbine and is being observed using SCADA systems.

3. Application of Vortex Generators to Wind Turbine Blades by Qing Tian, Dave Corson, Jonathon Baker

- The case study of an NREL 5-MW wind turbine demonstrated that total power loss per year due to the leading edge contamination can go up to 6.6%. It suggested that VGs can be used to improve wind turbine performance and result in 1.0% increase in annual energy production (AEP) by suppressing the flow separation and mitigating the soiling effect. Their mounting position along the blade span is important to the overall turbine performance.
- A validation study of numerical models is carried out and based on the OSU wind tunnel test cases for two different airfoils (NACA4415 and S814) at steady-state conditions.
- The NACA 4415 and S814 airfoil models were tested, under steady state and unsteady conditions, also with and without the leading edge contamination in the Ohio State University Aeronautical and Astronautical Research Laboratory 3x5 subsonic wind tunnel. The wind tunnel test results from above studies were used in the present study for comparison with the simulation results.

- The case study of an NREL 5-MW wind turbine indicated that surface roughness may play a part in large degradation of wind turbine power production, which to some extent may be mitigated by installation of VGs on wind turbine blades. Based on the study, it is not recommended to install vortex generators too far out on the blades, which may add extra drag and produce less AEP gain. The study also presented a better configuration of vortex generators, which were located in-between 15% and 41% of the blade span and resulted in an increase in AEP of 1%.
- According to the study, the leading edge grid roughness (LEGR) significantly decreased an airfoil performance, causing them to stall earlier by imposing extra drag. Vortex generators slow down stall and increase L_{max} , thus eliminating the unwanted consequences of the LEGR, to some extent. It is concluded that accurate predictions of aerodynamic coefficients for attached flow can be achieved with the current roughness model. However, the current turbulence model is not sufficient for accurate predictions in the post stall region. This is understandable and observed in the investigations by other researchers. As the flow is highly separated and unsteady around stall, a more complicated model such as LES or DES, is necessary in order to capture the relevant flow physics.

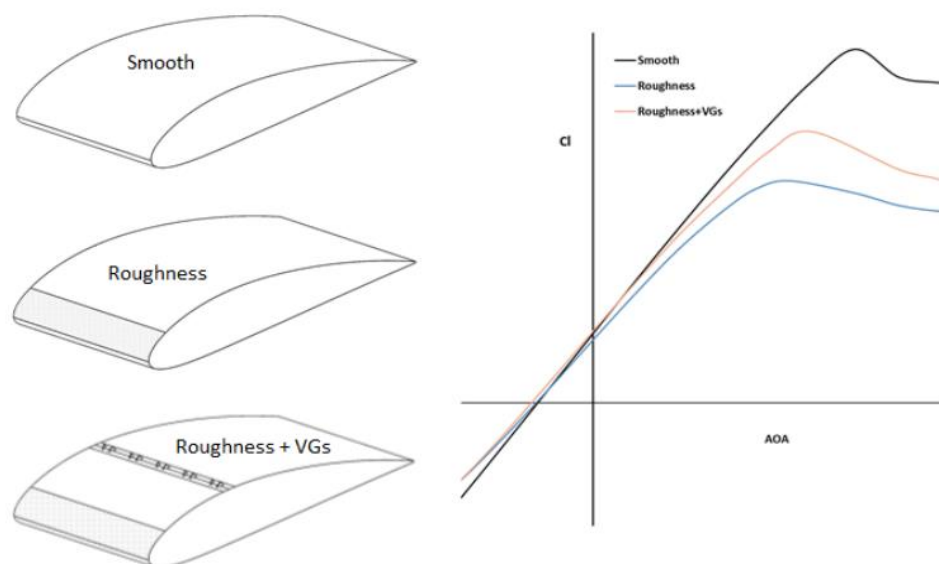


Fig.2.1. Aerodynamic impact to airfoil lift due to leading edge roughness and vortex generators

CHAPTER 3

AIM AND SCOPE

The main aim of the project is to improve the aerodynamic performance by placing an array of vortex generators on the wind turbine blade and to measure the lift and drag produced by the blade after CFD analysis for the NACA 0015 airfoil horizontal axis wind turbine blade.

In this investigation, we are going to design horizontal axis wind turbine blades of length 60 m using NACA 0015 airfoil in CATIA equipped with arrays of spike and dinoshell vortex generators. Then, importing the STL files to ANSYS FLUENT software for creating a computational domain and applying boundary conditions, here the load acting is pressure of the wind on the blade surface (velocity inlet and pressure outlet and 4-walls symmetry) and energy equation with flow velocity of 10m/s. Using K- ϵ standard wall function turbulence model (to simulate mean flow characteristics for turbulent flow conditions), CFD analysis has been done with meshing using the default size. Up to 500 iterations are performed and the result parameters are generated. The variation in velocity, pressure and turbulence pathlines have been observed by evaluating counters figures and plots. The focus is to find L and D, Cl and Cd values for single-sided VG blades and then to calculate the aerodynamic efficiency of both and declare the better one to get the power output and compare the results with that of a normal blade of similar length. The selected VG produced a max. vorticity of 55.7% more than the average from the LE and the lift coefficient was 0.42% higher compared to its average. The selected VG design and layout were adopted in a multi-MW wind turbine which reduced stall occurrence in the blade root area, as estimated by the simulation results. Fabrication of the blade with better aerodynamic performance (L/D ratio) is done and it is tested in the Open Circuit Low Speed Subsonic Wind Tunnel of test section size 600mmx600mm, later smoke flow visualization technique is also performed to capture streamline formation on the model surface instantaneously to study the flow separation. From the experimentation results, power generated is calculated and compared with that of the software analysis. The project comprised several wind tunnel tests as well as numerical investigations.

CHAPTER 4

METHODOLOGY

4.1. GRID DEVELOPMENT

The unstructured meshing software program in Ansys was used to discretize the fluid aspect geometry. The mesher operates without delay at the underlying CAD version to supply discretizations which might be true to the original geometry. A Delaunay triangulation approach is used to construct the surface mesh. The underlying CAD kernel is used to determine the location of nodes at the surfaces, ensuring that the discretization represents the CAD model as it should be, because of the high geometric component ratio of the airfoil section, meshing the floor with isotropic triangular elements proves to be a really inefficient approach. To treat this situation, and hold the advantage of fully automated unstructured meshing, we make use of dependent floor meshing along the span of the airfoil. The nodal distribution is clustered close to the main and trailing edges of the airfoil to capture the high tiers of curvature in these areas. This point distribution is then extruded inside the span wise path of the airfoil to create established, anisotropic surface factors, because of the complexity of the geometry, unstructured mesh with isotropic triangular elements are applied in the vicinity of vortex generators.

Upon completing the surface meshing process, extruding the elements on the airfoil in the surface normal direction creates elements that are clustered near the surface so that the steep near wall gradients in the boundary layer is resolved. The extrusion process is continued until the last layer of extruded elements has a similar height as what is specified for the surrounding volume elements. The final conformal mesh is produced after meshing and uniting the remaining volume with the existing surface and boundary layer mesh. Local volume refinement zones are used in the vicinity of the vortex generators, airfoil, and in the wake region immediately downstream of the airfoil to maintain a high level of resolution in those areas.

4.2. NUMERICAL METHODOLOGY

Numerical simulation has been carried using fluent software for modeling fluid flow and heat transfer in complex geometries. It provides complete mesh flexibility, including the

ability to solve the flow problems using unstructured meshes that can be generated about complex geometries with the relative ease. It is written in the C language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible.

All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface. The reference procedural steps for solving a problem in FLUENT include:

- 1) Define the modelling goals
- 2) Create the model geometry and grid
- 3) Set up the solver and physical models
- 4) Compute and monitor the solution
- 5) Examine and save the results

Consider revisions to the numerical or physical model parameters, if necessary.

Three major numerical analysis strategies, namely, direct numerical simulation (DNS), big eddy simulation (LES), and RANS simulation, may be used to analyze turbulent flow fields. To investigate the overall vortex behavior precisely in a turbulent flow field discipline containing vortices of diverse scales, the calculation grid size have to be smaller than the minimal space scale of the vortex structure and the time taken should be less than the minimal time scale of the vortex variation. LES uses the spatially averaged Navier-Stokes equation which immediately calculate vortices extra than the grid scale, not implying sub grid-scale version for vortices smaller than the grid scale. DNS and LES require huge resources for their calculations, making them unfeasible in realistic engineering applications. In contrast, RANS yields a decrease decision evaluation than DNS or LES; it's far extensively used in engineering applications as it does no longer require excessive-resolution calculation grids.

Turbulence models which can be extensively used for RANS numerical evaluation in engineering consist of $k-\epsilon$, $k-\omega$, and SST. The $k-\epsilon$ turbulence model exactly analyzes the turbulence conduct in free-movement areas with small strain gradients; however, it inaccurately estimates the boundary layer separation in viscous sub-layer areas. Although, the $k-\omega$ turbulence version, which was developed via Wilcox, accurately analyzes the

separation as a result of adverse stress gradients, it is sensitive to unfastened circulation region. Menter blended the benefits of the k- ϵ and k- ω models and proposed the SST turbulence version. K- ϵ has been very popular because they require less computer memory and CPU time than LES. Simulation of the wind turbine flow typically includes the usage of CFD to research the performance of the move-sectional airfoil of the blade, and the numerical analysis and experimental outcomes are often in agreement.

Energy Conservation Equation:

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = -\nabla \cdot \left(\sum_j h_j J_j \right) + S_h$$

4.2.1 Turbulence Models

Fluctuating velocity fields characterize turbulent flows. Transported quantities such as momentum, energy, and species concentration are mixed with these fluctuations and which causes them to fluctuate as well, resulting in small scale but high frequency fluctuations which are too computationally expensive to simulate directly in practical engineering applications. Instead, the exact governing equations can be time-averaged or otherwise manipulated to remove the small scales, giving modified set of equations that are computationally less expensive to solve and which contain additional unknown variables, and to determine these variables in terms of unknown quantities, turbulence models are needed.

K- ϵ models-:

Standard k- ϵ model, Renormalization group (RNG) k- ϵ model, Realizable k- ϵ model

4.2.2. Choosing a Turbulence model

The choice of turbulence model will depend on parameters like physics behind the flow, the class of problem, the level of accuracy required, the available computational resources, and the time taken for simulation.

4.2.3. STANDARD K- ϵ MODEL

The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In the derivation of the k-ε model, the flow is assumed to be fully turbulent, and the effects of molecular viscosity can be ignored. The standard k-ε model is therefore valid only for fully turbulent flows.

Transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k$$

and

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon$$

4.2.4. Boundary Plane:

The boundary plane is set keeping in mind the axis-symmetric features. The five boundary conditions set in this modeling are as follows:

- Velocity Inlet Boundary Conditions
- Pressure Outlet Boundary Conditions
- Wall Boundary Conditions
- Symmetry Boundary Conditions
- Axis Boundary Conditions
 - Velocity Inlet Boundary Conditions:

The domain inlet wall is set as velocity inlet boundary. Velocity inlet boundary conditions are used to define the flow velocity, along with the relevant scalar properties of the flow, the flow inlets. The total (or Stagnation) properties of the flow are not fixed, so they will rise to whatever value is necessary to provide the prescribed velocity distribution.

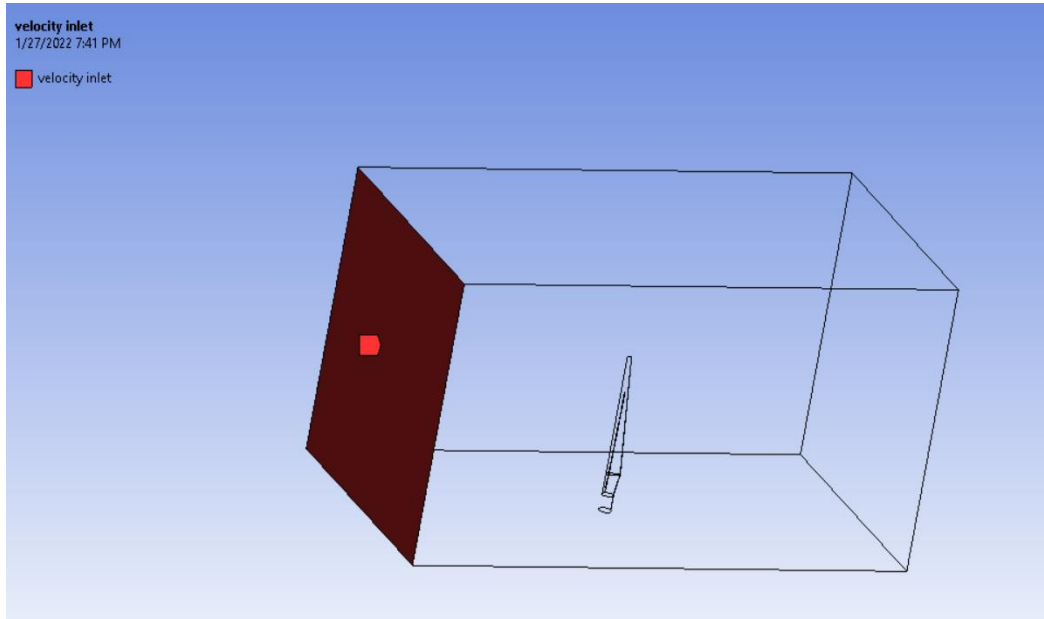


Fig.4.1. Velocity Inlet

- Pressure Outlet Boundary Conditions:

The domain outlet wall is set as a Pressure outlet boundary. Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. The value of static pressure specified is used only while the flow is subsonic.

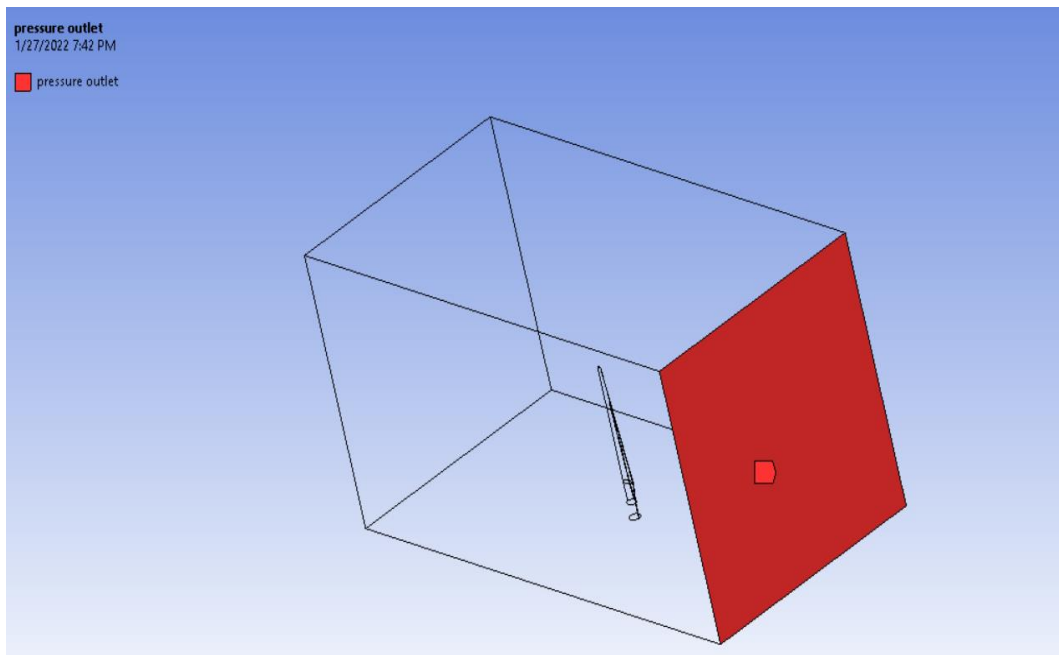


Fig.4.2. Pressure Outlet

- Wall Boundary Conditions:

The model wall is set as wall boundary. Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary conditions is enforced at walls by default, but it can be specified that a tangential velocity component in terms of translational or rotational motion of the wall boundary ,or model a slip wall by specifying shear. The shear stress and the heat transfer between the fluid and wall are computed based on the flow details in the local flow field.

- Axis Boundary Conditions:

The centre axis of the axis-symmetric model is set as axis boundary. The axis boundary type must be used as the centerline of an axis-symmetric geometry. It can also be used for the centerline of a cylindrical-polar quadrilateral or hexahedral grid. No Boundary conditions are required at axis boundaries.

- Meshing:

Meshing is the procedure of dividing a continuous unit into smaller character gadgets in computational domain.

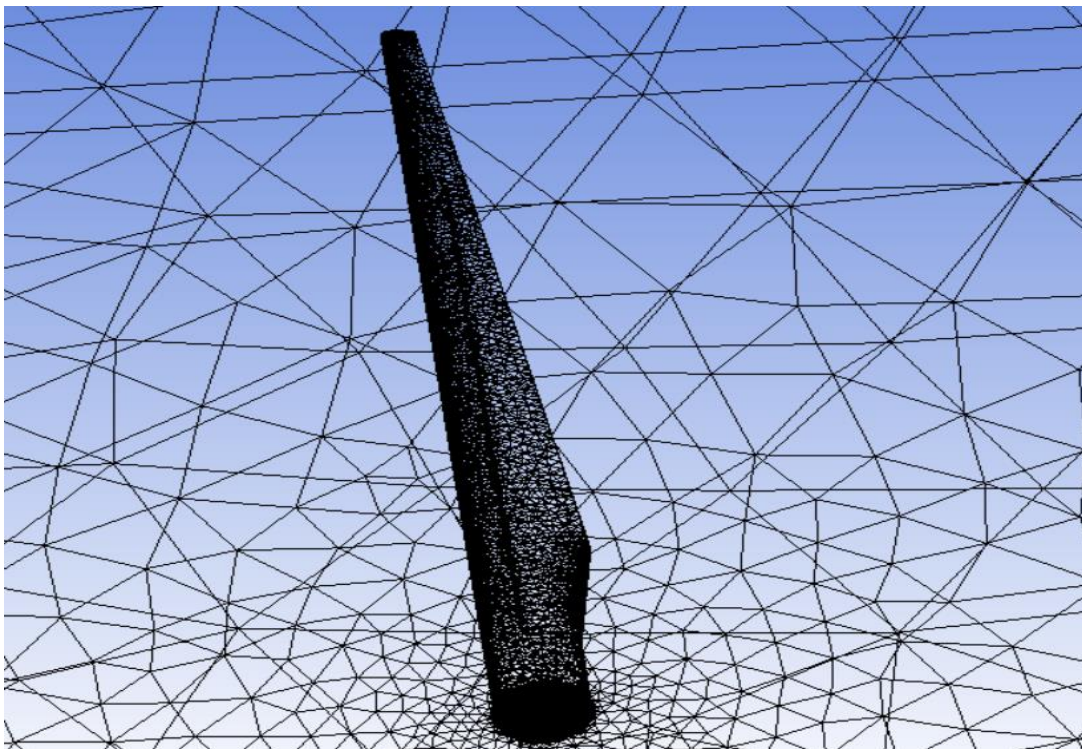


Fig.4.3.Meshing

4.2.5. Solver and Material Selection and Operating Condition Defining:

Solver: The solver is defined first Energy equation is taken into consideration. The viscous medium is also taken. First the analysis is carried using turbulent flow with k- ϵ model, specifically realizable.

The following conditions are taken in solver:

TABLE 4.1: Solver Specifications

Solver	Pressure Based
Formulation	Implicit
Time	Unsteady
Space	Axisymmetric
Velocity Formulation	Absolute
Porous Formulation	Physical Velocity
Gradient options	Green-Gauss Cell
Viscous Model	k- ϵ

Selection of Material and Operating Conditions:

Material selected is air. The Properties of the fluid are as follows.

The following values are entered in Viscous and Material Selection.

TABLE 4.2: Material Properties

Material Type	Fluid
Fluid	Air
Density	1.225 kg/m ³
Viscosity	1.7894 x 10 ⁻⁵ kg/ms
Pressure	101325 Pa

4.2.6. Boundary conditions:

The boundary conditions selected are as follows:

- Velocity Inlet Boundary Conditions for the domain inlet.
- Pressure Outlet Boundary Conditions for the domain outlet.
- Wall Boundary Conditions for the model wall.
- Symmetry Boundary Conditions for the upper edge of the domain.
- Axis Boundary Conditions for the bottom edge of the domain.

The velocity inlet conditions set are as tabulated below.

TABLE 4.3: Velocity Inlet and Pressure Outlet Conditions:

Inlet	Velocity Inlet ,Velocity =10 m/s
Specific method	Intensity & Viscosity ratio
Turbulent Intensity	0.1
Viscosity ratio	10
Outlet	Pressure Outlet
Specific method	Intensity & Viscosity ratio
Turbulent Intensity	0.1
Viscosity ratio	10

4.2.7. Report Reference Values

In reporting parameters, the reference values are given as following:

TABLE 4.4: Report Reference Values

Density-1.225 kg/m ³	Temperature-288.16 K
Enthalpy-0 J/kg	Velocity-10 m/s
Length-1 m	Viscosity-0.17894 x 10 ⁻⁴ kg/ms
Pressure-0 Pa	Reference Zone-Fluid

CHAPTER 5

UNDERSTANDING WORKFLOW with DESIGN AND CFD ANALYSIS

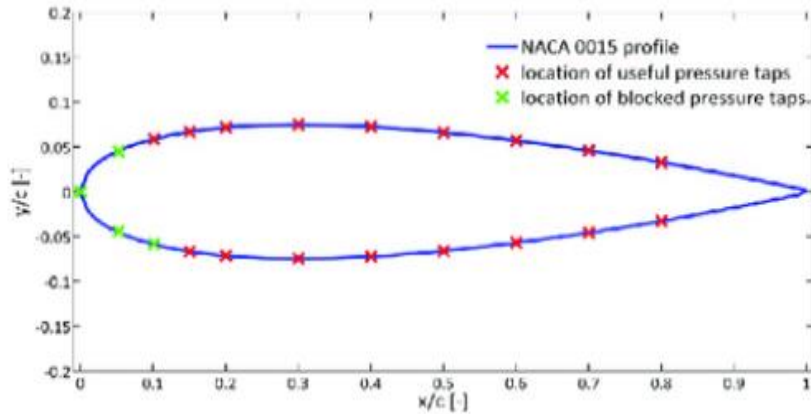


Fig.5.1. NACA 0015 airfoil

Considering the blade aerodynamics and its structural configurations, we have chosen NACA 0015 airfoil that is further extended in the spanwise direction. It is a symmetrical airfoil with a 15% thickness to chord (t/c) ratio, that are used in many applications such as aircraft vertical stabilizers, wind turbines, submarine fins, rotary, fixed wings, etc. A 2D wing section is analyzed initially at low speeds for lift, drag and moment characteristics.

Wind turbine towers and foundations must be designed to withstand heavy loads and moments due to extreme wind conditions to prevent failures, as well as other forces that are introduced with alternative site designs. The forces that the tower and foundation must resist are wind loads, ice loads, and the self-weight of the tower. The tower structure must also resist earthquake loads, which can be designed based on checking resistance in the steel's plastic range. In addition, the soil has to have adequate bearing capacity to resist the loads on the tower and weight of the foundation. Wind turbines are also classified by the amount of power that they can generate, and are directly correlated to the physical size of the turbine (larger wind turbines produce more energy). The three types of turbines in terms of power output levels are utility-, industrial-, and residential-scaled wind turbines.

Table 5.1.Wind Turbine Specification

Category	Value
No. of Blades	3
Rotor Diameter	110 m
Rated wind speed	10 m/s
Rotational speed	Variable
Rotational Path	Clockwise
Airfoil profile	NACA 0015
Target Wind Class	I
Rated Wind Power	4 MW
Simulation Region	Volume

Table 5.2.Specifications of VG w.r.t chord length

Number of VGs per blade	45
Where are they placed	At 10~30% of blade length from the root, 20% of chord length from LE
Layout	1.5%
Length	1.5%
Height	1%
Angle measured with respect to direction of the chord	11°

5.1. SINGLE-SIDED SPIKE VORTEX GENERATOR BLADE

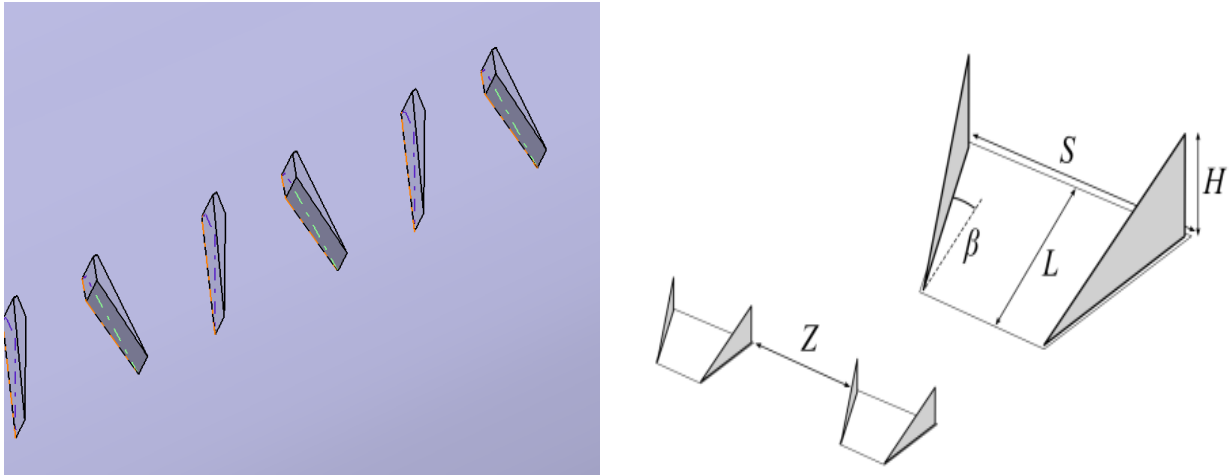


Fig.5.2. Array of Spike VGs on a HAWT Blade

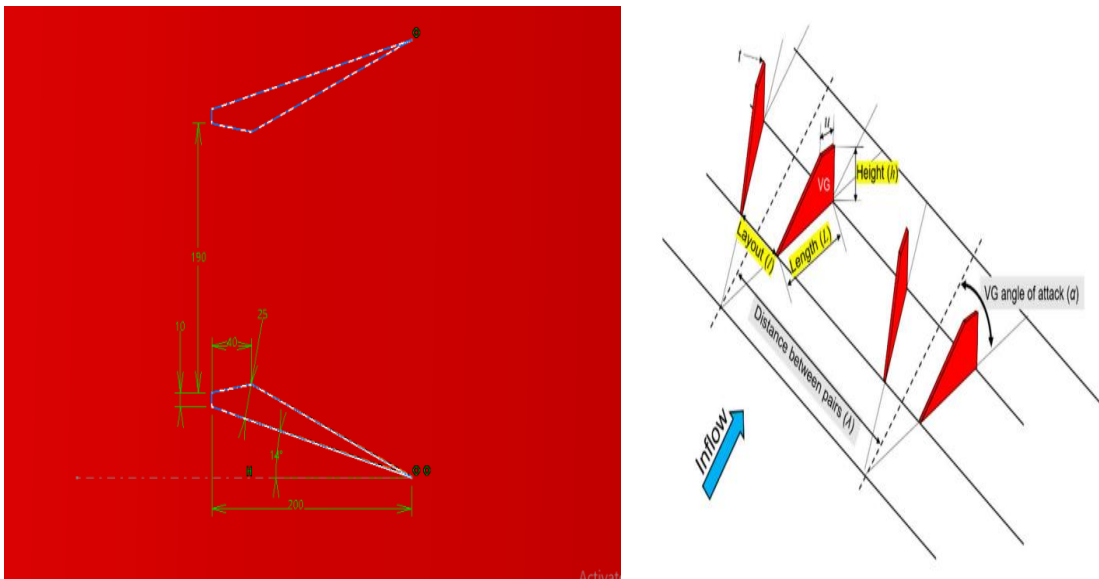


Fig.5.3. Thickness = 250mm Dimension of Spike vortex generator.

Here, the layout (l) = 190mm, length (L) = 200mm, $\beta = 14^\circ$, $H = 10$ mm, spacing, $Z = 300$ mm, and angle of attack is 11° . There are 45 spike pairs on a 60 m HAWT blade. Z/H is recommended to be 5 according to previous studies. It can be seen the layout and shape of the VGs affect the overall power output efficiency. They are placed at $0.3c$ on the suction surface at an angle of 15° to the incoming flow for the sake of producing pairs of counter-rotating vortices.

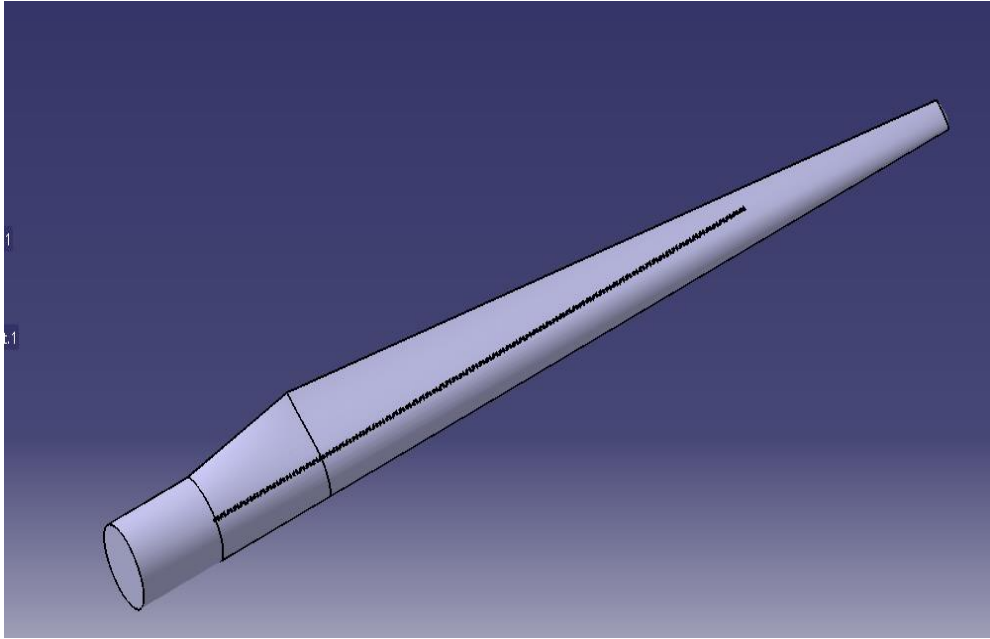


Fig.5.4. Horizontal axis wind turbine blade with Spike vortex generators designed in CATIA

We designed wind turbine blade in CATIA V5 and introduced VGs to increase lift. As the air flows over the surface which will disturb the air flow over the surface, this disturbance will help increasing the lift.

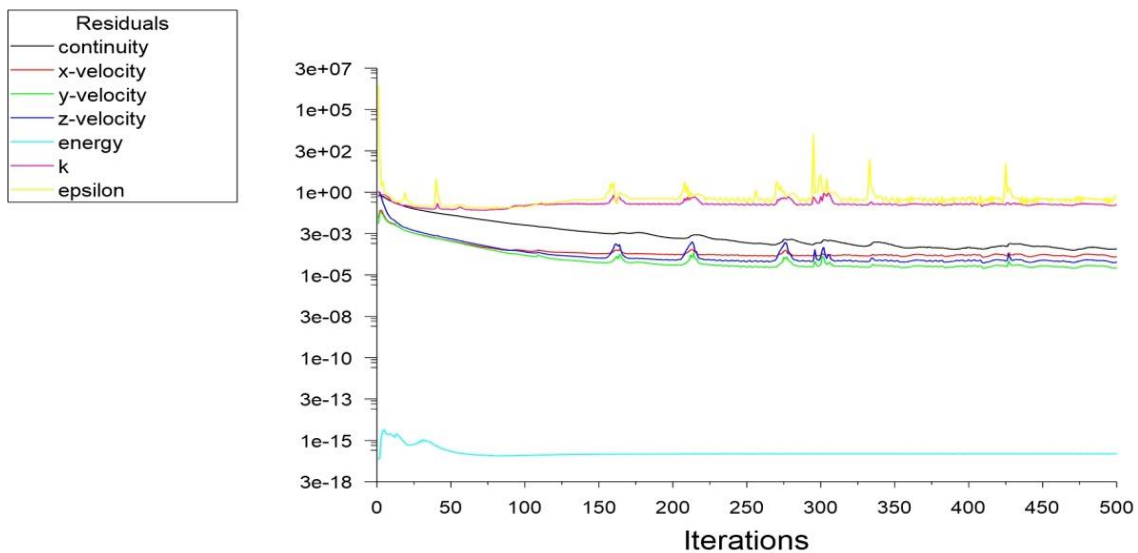


Fig.5.5. Scaled Residuals Graph Plotted for 500 iterations using Ansys for SPIKE VGs Blade

This graph shows us the scaled residual plots iterations against boundary for the blade with spike VGs installed on it.

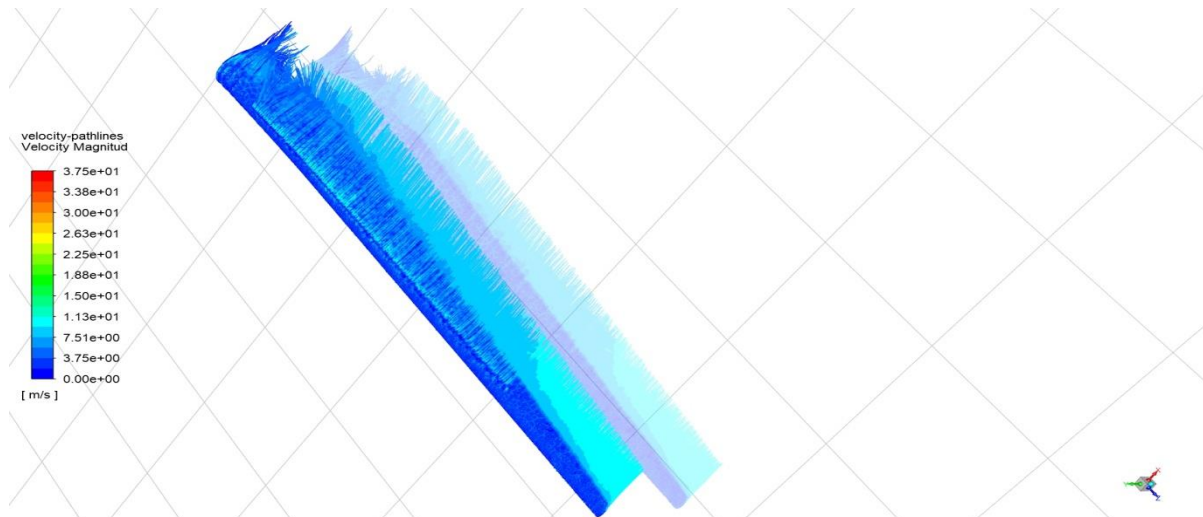


Fig.5.6. Velocity Pathlines for single-sided spike VGs installed HAWT Blade

How velocity changes along the motion of a particle of fluid flowing over the blade. They help to visualize the flow. Also a method of measuring stream wise discharge by which point velocity measurements are converted into average cross-sectional flow velocities by contouring the point.

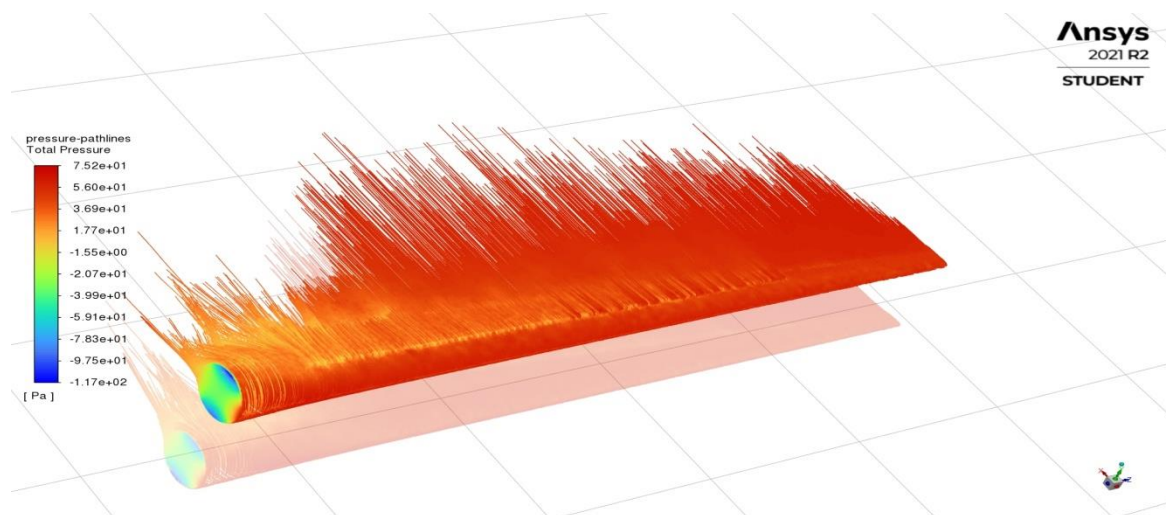


Fig.5.7. Pressure Pathlines for single-sided spike VGs installed HAWT Blade

How pressure changes along the motion of a particle of fluid flowing over the blade. They help to visualize the flow. Also a method of measuring stream wise discharge by which point pressure measurements are converted into average cross-sectional flow pressures by contouring the point.

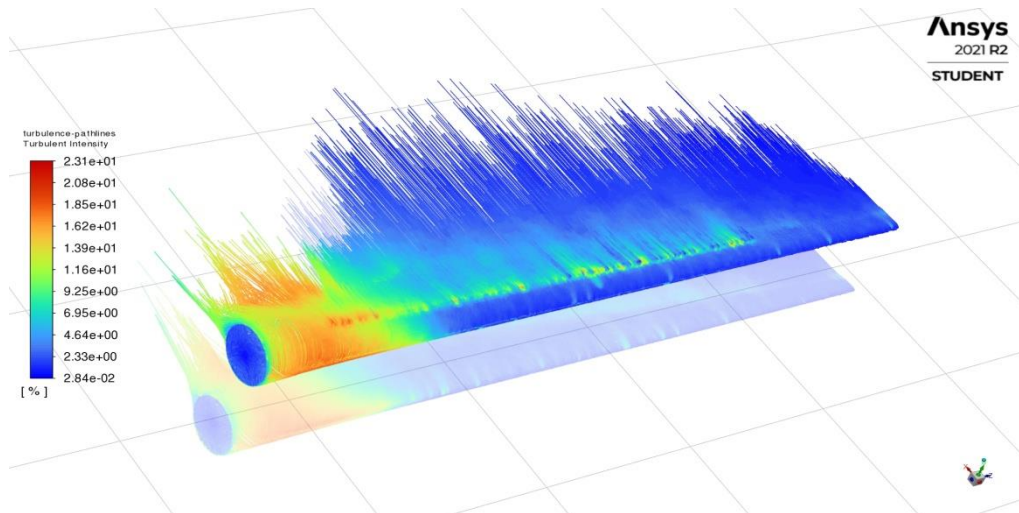


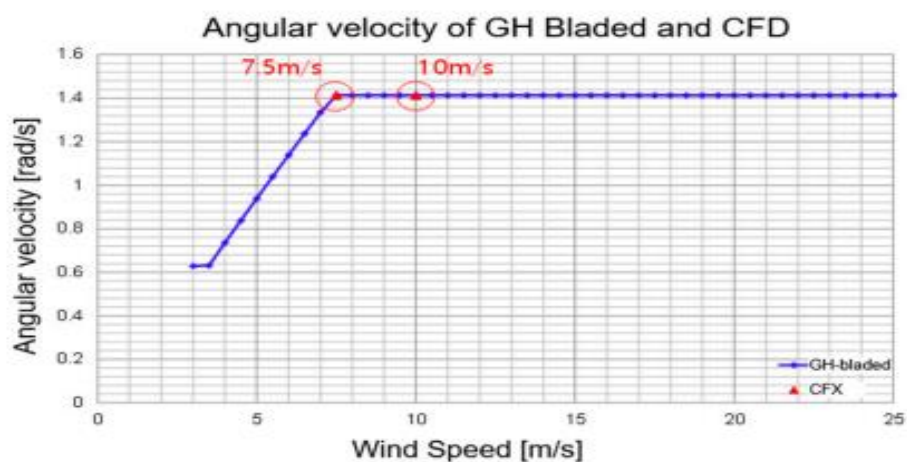
Fig.5.8. Turbulence Pathlines for single-sided spike VGs installed HAWT Blade

To know how much unsteady movement of fluid particles is occurring over the blade surface. They help to visualize the flow.

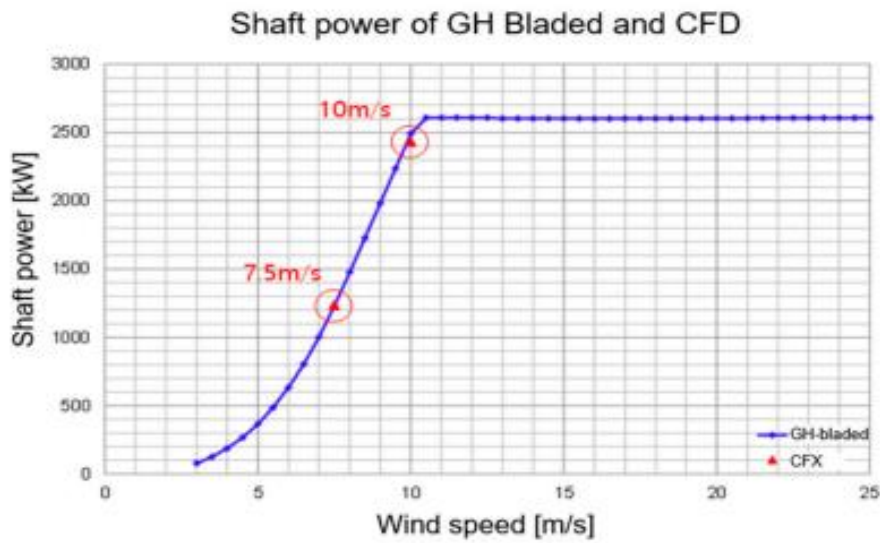
lift-force	
Lift	(n)
wall-spike_blade	1134.6686

Fig.5.9. Computational Report of aerodynamic parameters of spike VGs HAWT Blade

Lift can be converted to energy to understand how much power can be generated.



(a) Angular Velocity vs Wind Speed



(b) Shaft Power vs Wind Speed

Fig.5.10. Comparison of GH Bladed Design Data and CFD Results

It can be seen that both angular velocity (ω) and shaft power (P) vary linearly with wind speed up to a certain limit which is 7m/s in case of ω and 10-15m/s in case of P and then become constant with further increase in speed.

Case	Wind Speed (m/s)	Angular Velocity(rad/s)	GH-Bladed Power(kW)	Power from analysis(kW)	Note
1	7.5	1.414	2231.09	2234.69	0.236%
2	10.0	1.414	4290.51	4231.87	1.459%

Table 5.3.BEMT Design Data and CFD Results

This table compares the blade's CFD analysis results with the GH Bladed share software (commercial code for design of wind turbines) design values. The analysis results can be proved to be steady with an error margin of 0.236% and 1.459% at wind speeds of 7.5m/s and 10 m/s respectively. The BEMT based analyses were also conducted using GH Bladed.

The below figure shows that the velocity contour along the chord of the cross-sectional blade at AoA 12° with and without VGs.

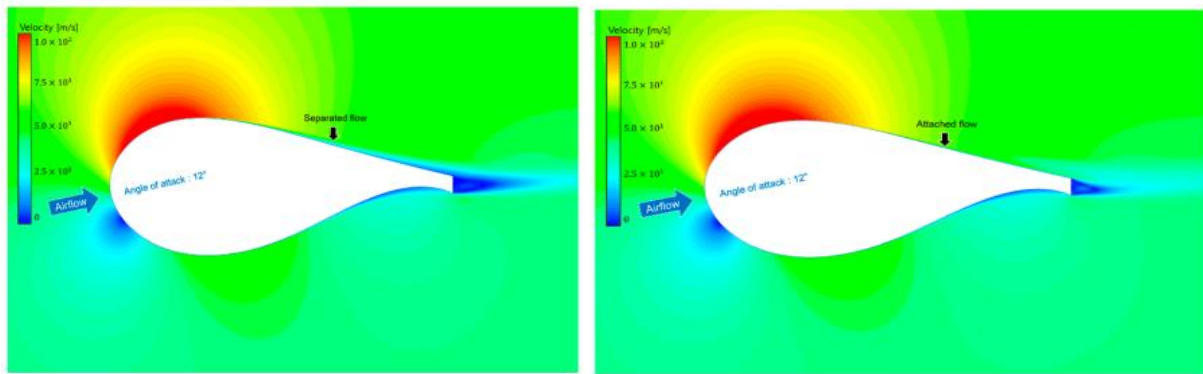


Fig.5.11. Cross-sectional velocity contour of the blade: (a) without VG; (b) with VG.

From this we can observe that in case of (a), the flow separation takes place and the fluid does not accelerate along the suction surface, but in case of (b), flow separation does not occur and then it is attached to the suction surface along which the flow accelerates.

A low pressure region is generated as wake, when flow separation takes place on the suction surface, which further increases C_d and decreases C_l . As shown below, the airfoil feels a higher lift at 5.9° AoA or more and a lower drag at 14.8° AoA or more, when the VG is present.

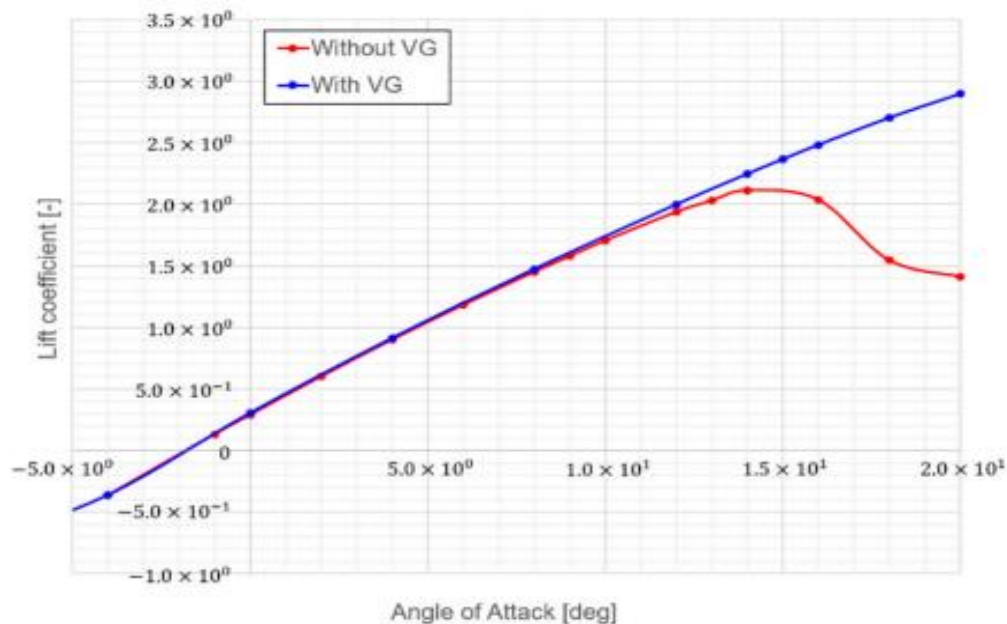


Fig.5.12. (a) Lift Coefficient vs. Angle of Attack

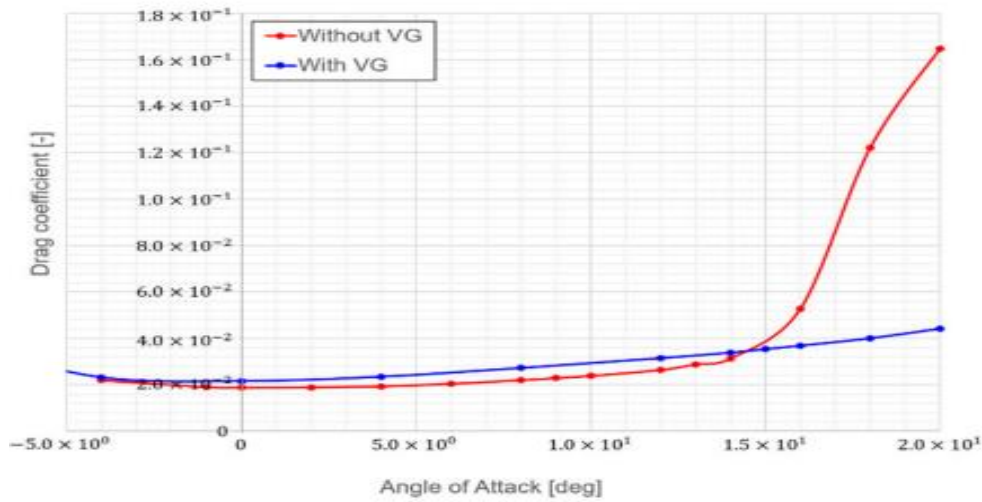


Fig.5.12 (b) Drag Coefficient, C_d vs. Angle of Attack, AoA

5.1.1 Study of the influence of VG design parameters:

C_L , C_D , and the maximum vorticity, ω at the positions 50% and 100% from the LE to the TE, were studied. The RSM used a Locally Weighted Scatterplot Smoother second equation to find pairs that could show the best VG effect using the 3 independent design parameters, namely, layout (L), length (l), and height (h).

Table 5.4. Case studies results to select the VG design parameters.

Case	C_l	C_d	Maximum vorticity at position in the direction of the chord(1/s)			
			0.25	0.50	0.75	1.0
01	2.025	0.0265	10363	3078	3033	1625
02	2.010	0.0302	10289	3370	3054	1766
03	1.974	0.0627	13641	3773	2535	1200
04	2.013	0.0293	13554	3245	3472	1154
05	1.990	0.0327	16308	3712	2374	975
06	1.978	0.0344	14352	3912	2449	1128

07	2.014	0.0291	13926	3205	2335	1128
08	1.996	0.0319	16066	3698	2338	1101
09	1.978	0.0348	15392	3675	2233	1112
10	2.113	0.0289	12809	3092	2497	1205
11	2.001	0.0315	13846	3471	2446	1061
12	1.991	0.0332	12298	2876	1965	969
13	1.993	0.0306	13626	2890	1859	830
14	2.004	0.0311	14949	3463	2423	1141
15	1.854	0.0220	12029	2490	1392	674
16	2.019	0.0287	14391	3001	2391	1209
17	2.004	0.0314	16751	3452	2255	1117
18	2.000	0.0309	16487	3616	2237	1071
19	2.019	0.0285	13107	2865	2409	1147
20	2.008	0.0309	14943	3211	2177	1111
21	2.002	0.0322	12821	2622	1704	955
22	2.021	0.0283	13380	3134	2404	1235
23	2.010	0.0307	15369	3498	2174	1196
24	1.996	0.0331	16024	3433	2104	1097
25	2.020	0.0282	13507	3228	2336	1266
26	2.009	0.0308	16087	3596	2106	1195
27	2.002	0.0325	15635	2373	1614	966

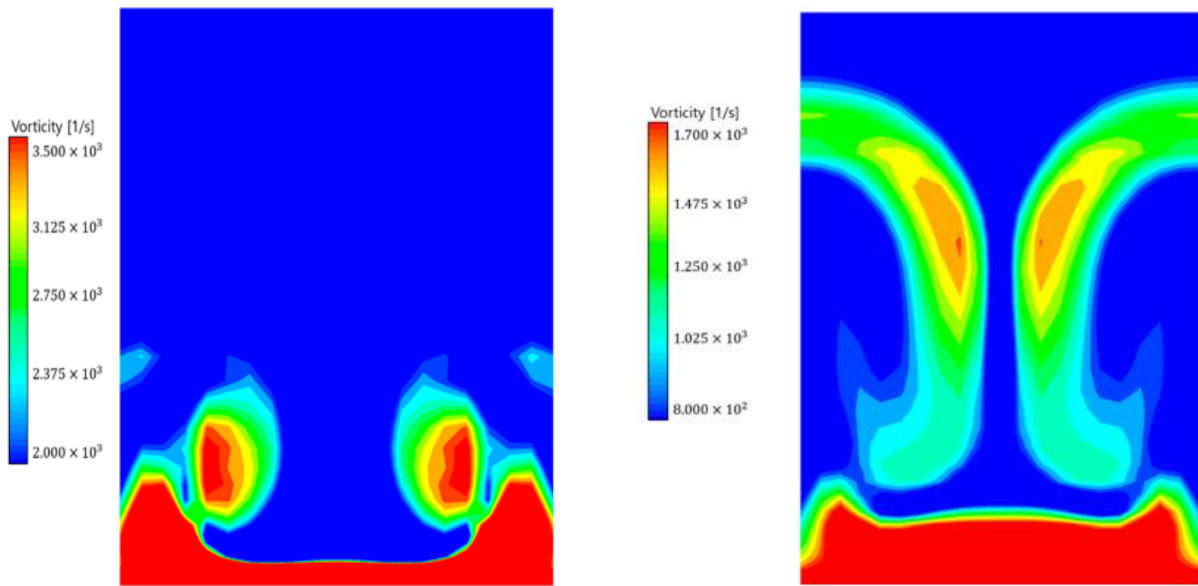


Fig.5.13. Vorticity contours at the (a) midpoint of the chord and (b) Trailing edge

The vorticity contours at locations 0.5c and 1.0c from the LE are represented for cases 3 and 20 respectively.

5.2. SINGLE-SIDED DINOSHELL VORTEX GENERATOR

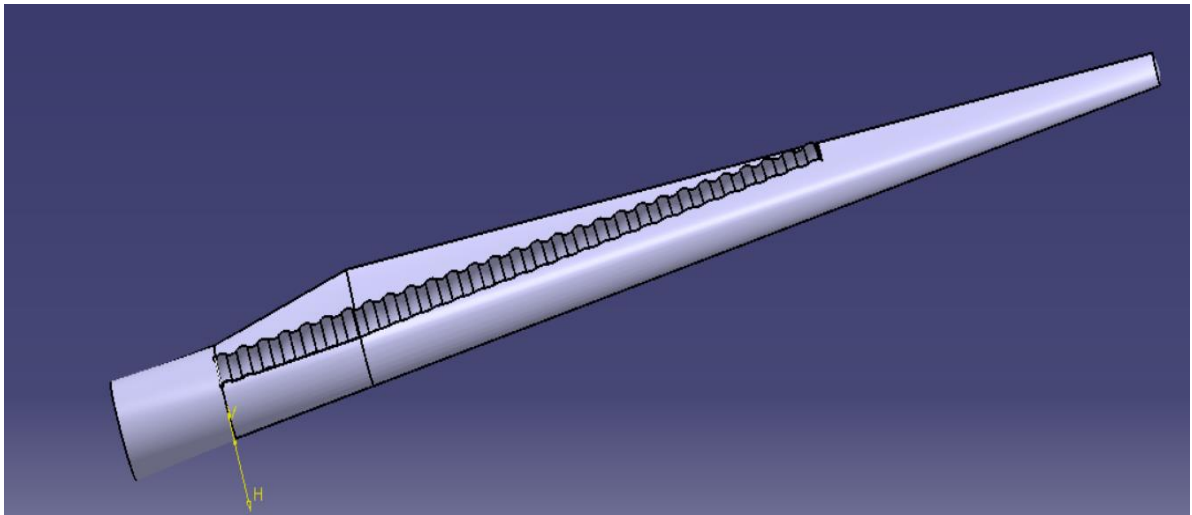


Fig.5.14. HAWT Blade with Dinoshell Vortex Generators

Dinoshell VGs are curved and placed laterally to the chord-wise section of the blade.

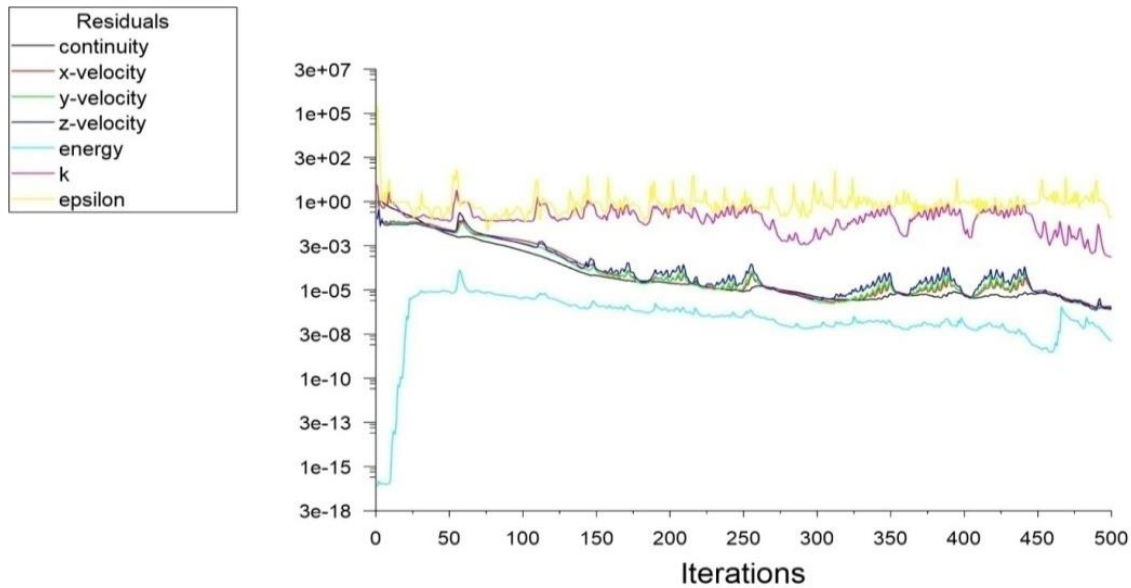


Fig.5.15. Scaled Residuals Graph Plotted for 500 iterations using Ansys for Dinoshell VGs Blade

This graph shows us the scaled residual plots iterations against boundary for the blade with dinoshell VGs installed on it.

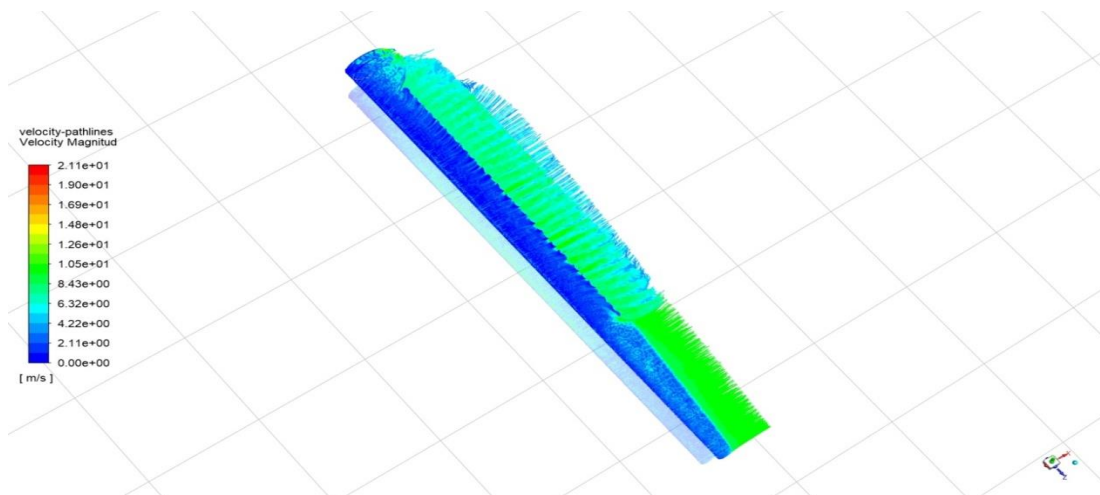


Fig.5.16. Velocity Pathlines for single-sided dinoshell VGs installed HAWT Blade

How velocity changes along the motion of a particle of fluid flowing over the blade. They help to visualize the flow. Also a method of measuring stream wise discharge by which point velocity measurements are converted into average cross-sectional flow velocities by contouring the point.

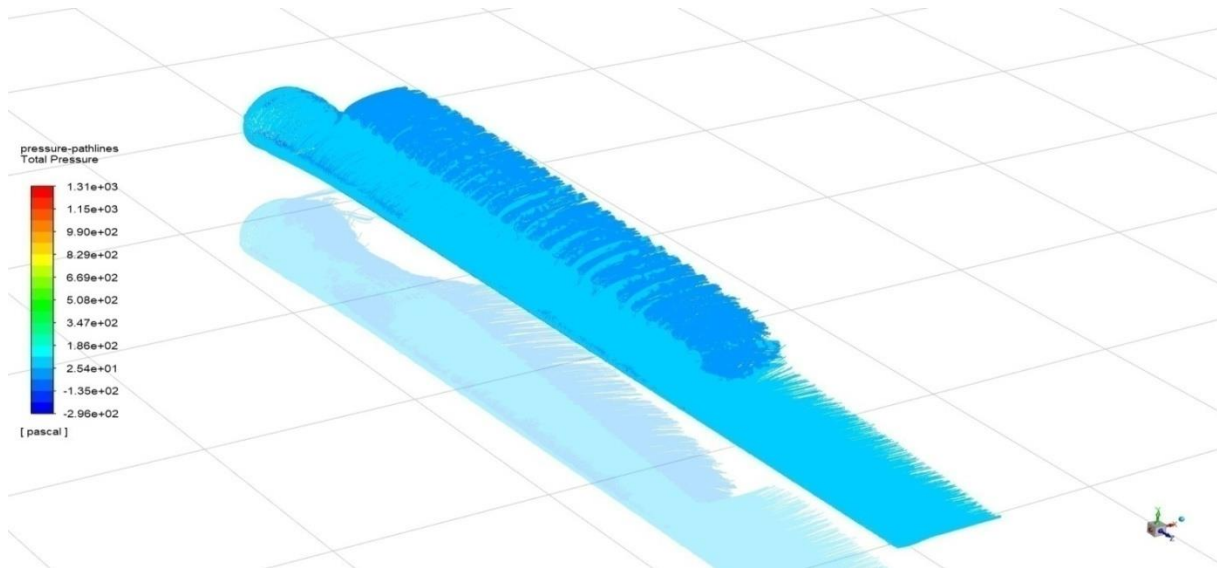


Fig.5.17. Pressure Pathlines for single-sided dinoshell VGs installed HAWT Blade

How pressure changes along the motion of a particle of fluid flowing over the blade. They help to visualize the flow. Also a method of measuring stream wise discharge by which point pressure measurements are converted into average cross-sectional flow pressures by contouring the point.

5.3. NORMAL HORIZONTAL AXIS WIND TURBINE BLADE

A HAWT has a comparable layout to a wind turbine; it has blades that seem like a propeller spinning on the horizontal axis. Horizontal axis wind turbines have the number one rotor shaft and electric powered generator at the pinnacle of a tower, and they have to be pointed into the wind. Small turbines are pointed in a manner of a smooth wind vane placed perpendicular with the rotor (blades), while big generators usually use a wind sensor coupled with a servo motor to expose the turbine into the wind. Most big wind turbines have a gearbox, which turns the slow rotation of the rotor right into a faster rotation that is more appropriate to pressure an electrical generator.

TRADITIONAL HAWT ADVANTAGES:

- Strong assisting tower is now no longer wished due to the fact generator, gearbox and different additives are positioned on the ground.
- Low manufacturing values as compared to horizontal axis wind turbines.

- As there can be no want of pointing turbine in wind course to be green so yaw pressure and pitch mechanism isn't wished.
- Smooth set up as compared to different wind turbine.
- Easy to transport from one region to different.
- Low renovation charges.
- They can be installed in town regions.
- Low threat for human and birds because of the truth blades moves at notably low speeds.
- They are in particular appropriate for regions with immoderate weather conditions, like within side the mountains in which they are able to deliver strength to mountain huts.

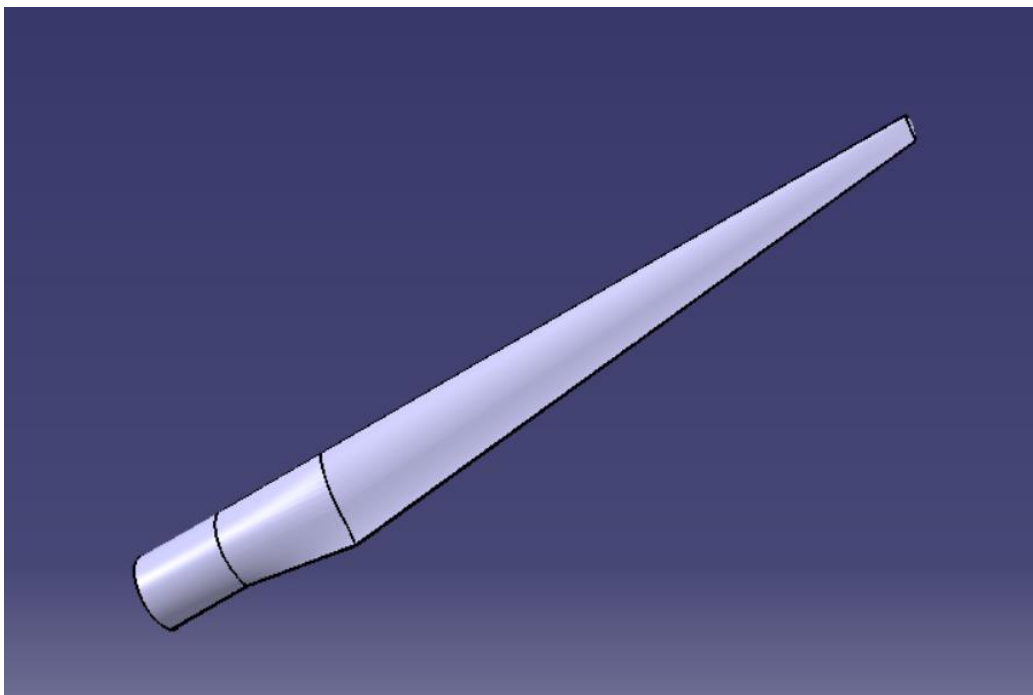


Fig.5.18. Normal HAWT Blade of length 60 m

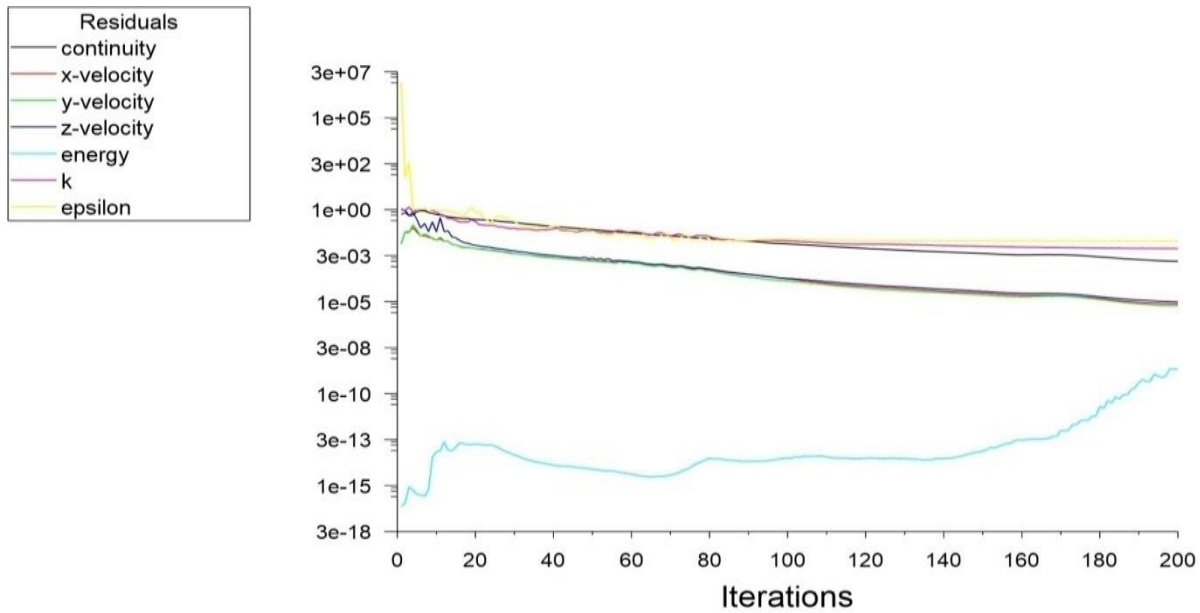


Fig.5.19. Scaled Residuals Graph Plotted for 200 iterations using Ansys for Normal HAWT Blade

This graph shows us the scaled residual plots iterations against boundary for the blade.

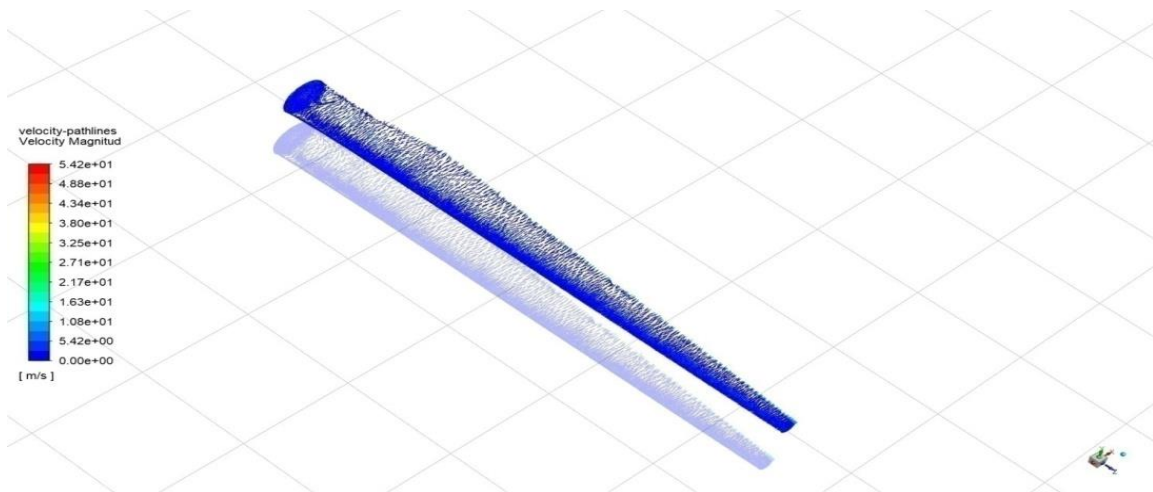


Fig.5.20. Velocity Pathlines for normal HAWT Blade

How velocity changes along the motion of a particle of fluid flowing over the blade. They help to visualize the flow. Also a method of measuring stream wise discharge by which point velocity measurements are converted into average cross-sectional flow velocities by contouring the point.

CHAPTER 6

RESULTS AND DISCUSSION

Hence, from the above data, we can conclude that Spike vortex generator Blade have better aerodynamic performance compared to Dinoshell Vortex Generator Blade because it creates low pressure region. If air crosses the spike vortex generator, it strikes the shape and splits and moves. This effect creates vacuum or low pressure region that sucks more air means more incoming air flow, as a result velocity is increased and lift is also increased.

CALCULATIONS:

The area swept by the turbine blades in motion can be calculated from the length of the turbine blades using the equation for the area of a circle: $A = \pi r^2$.

GIVEN: Blade length, $l=60$ m

Wind Speed, $V=10$ m/s

Air Density, $\rho=1.225$ kg/m³

Power Coefficient, $C_p=0.593$

Inserting the value for blade length as the radius of the swept area into equation we have:

$$l = r = 60\text{m}$$

$$A = \pi r^2 = \pi \times 60 \times 60 \text{ m}^2 = 11,310 \text{ m}^2$$

We can then calculate the power obtained after conversion from the wind into rotational energy in the turbine using equation:

$$P_{\text{avail}} = 0.5 \cdot \rho A V^3 C_p$$

$$P_{\text{avail}} = 0.5 \cdot 1.225 \cdot 11310 \cdot (10)^3 \cdot 0.593 \text{ W}$$

$$P_{\text{avail}} = 4.1078 \times 10^6 \text{ W} = 4.1 \text{ MW}$$

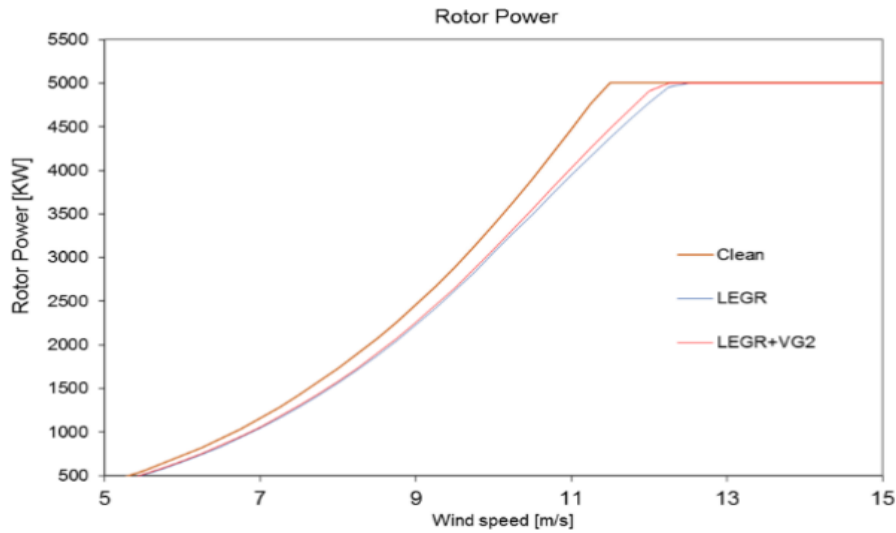


Fig.6.1. Power Curves of a typical 5MW wind turbine

This value is defined by the turbine designers although it is important to understand the relationship between all of these factors and to use this equation to calculate the power at wind speeds other than the rated wind speed. The shaft power is calculated as the product of torque and angular velocity of the simulation results. The calculated shaft power without and with the VG-mounted blade was approximately 2300 kW and 2450 kW, respectively, almost 2.50% more in the shaft power compared with the result before installation.

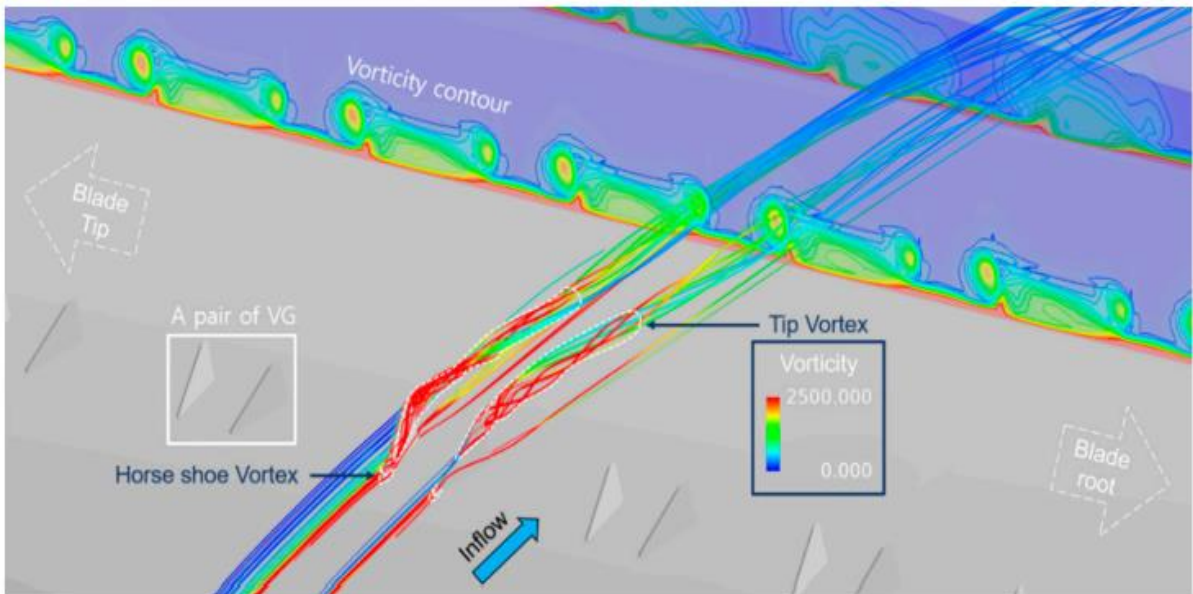


Fig.6.2. Flow characteristics near VG and vortex formed

The CFD evaluation of the viscous sub-layer was exactly executed through composing the entire grid with a hexahedron. The floor grid of the VG-set up blade used for the simulation, and the variety of grids is supplied. A couple of VGs each generates a vortex in the contrary direction and an excessive-power fluid flows into the boundary layer area of the low-electricity float area. The evaluation showed that a tip vortex generated on the tip of the VG and a vortex in the shape of a horseshoe is formed at the VG's lower front. The float separation is highly reduced on the suction side in comparison to the baseline blade. The VG paperwork a vortex and pushes the separation fluid to a distance of 8 m from the root, where waft separation is dominant. The shaft energy turned into calculated by means of multiplying the torque and angular velocity of the simulation results.

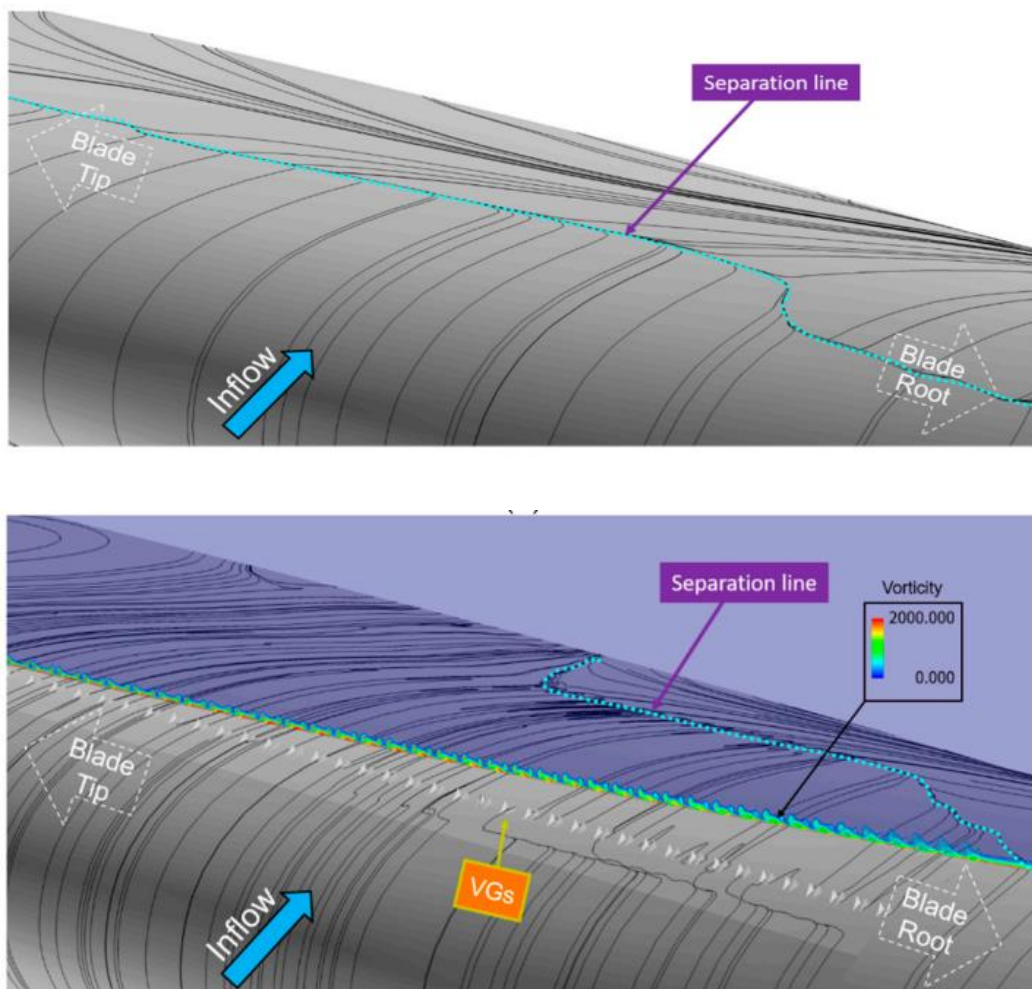


Fig.6.3. Aerodynamic effects on blade (a) without VGs (b) with VGs

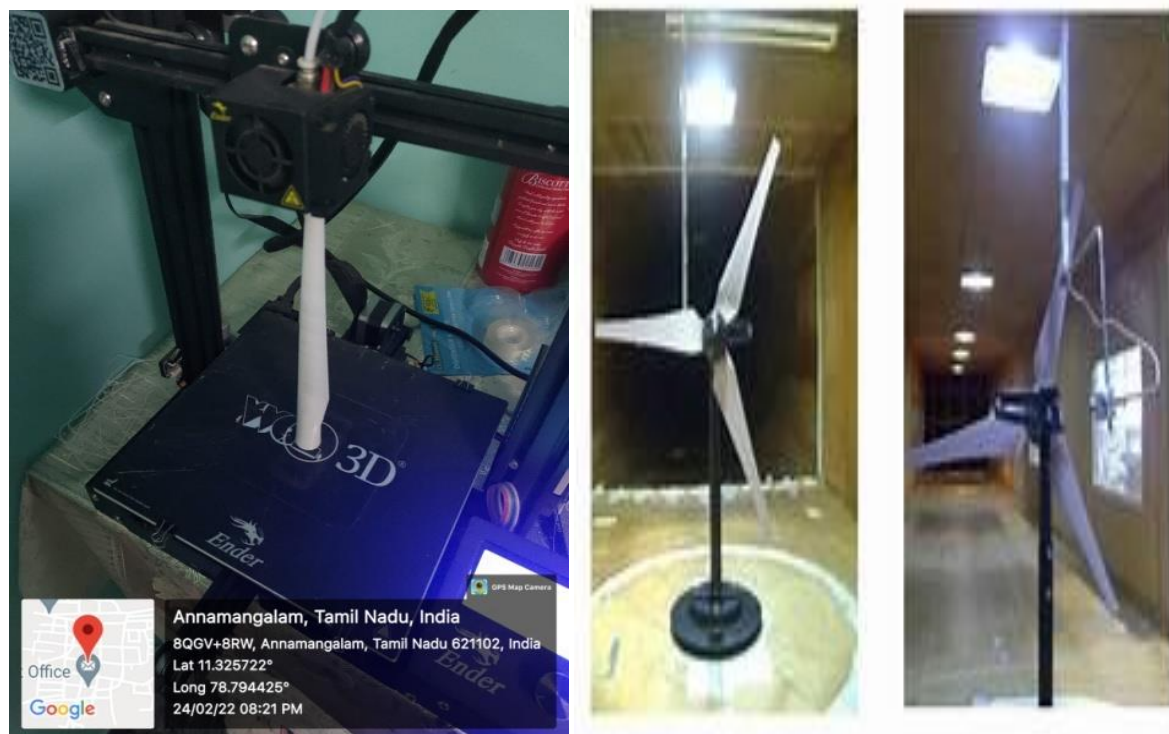


Fig.6.4.Fabrication and Experimentation

Fabrication of the blade with better aerodynamic performance (L/D ratio) is done using PLA (Polylactic Acid) Plastic which is a vegetable-based plastic material, with raw material as cornstarch. It is tested in the Open Circuit Low Speed Subsonic Wind Tunnel of test section size 600mmx600mm; later smoke flow visualization technique is also performed to capture streamline formation on the model surface instantaneously to study the flow separation. From the experimentation results, power generated is calculated and compared with that of the software analysis. The project comprised several wind tunnel tests as well as numerical investigations.

We insert 2mm hollow tubes into the 12 tappings made around the model in clock wise direction. 6 on the top surface and 6 on the bottom surface. The model with the tube is fitted to the wind tunnel and flow over the surface is studied at different RPM.

Condition of the wind tunnel

- Speed- 100,200,300 RPM
- Manometer liquid - Ethanol

Coefficient of Pressure (C_p) values are calculated using $C_p = (P_{Si} - P_s) / (P_t - P_s)$;

where P_s =static pressure and P_t = total pressure.

Experimental results are as follow:

Table 6.1.Experimental Readings

Ports/RPM	1	2	3	4	5	6	7	8	9	10	11	12
100	0.8	0.9	0.9	0.9	0.8	0.9	0.9	0.8	0.9	0.9	0.8	0.9
200	1.2	1.0	1.1	1.2	0.9	1.1	0.8	1.2	0.9	0.8	1.2	1.1
300	1.3	1.2	1.1	1.3	1.2	1.1	1.4	1.2	1.3	1.2	1.1	1.4

Table 6.2. C_p Port Readings

C_p / Ports	1	2	3	4	5	6	7	8	9	10	11	12
	2	2	2	2	0.3	2	2	2	2	2	2	2
	1	0	0	-1	2	3	1	1	2	0	1	1
	-0.33	0.33	-1	0	0.3	-0.33	-0.33	0.33	0	-1	-0.33	-0.33

- Calculate x/c and y/c for each port location

Further the x/c which is the distance from one tapping to the other in horizontal direction is measured from the fabricated model and y/c is the distance from each tapping to the chord

line in vertical direction. C_l , C_d will be calculated using the x/c and y/c values by choosing only the max. Negative value and min. negative value at every angle of attack.

$$C_l = (\text{max value upper camber} - \text{min value lower camber}) * (x/c_{\text{max}} - x/c_{\text{min}})$$

$$C_d = (\text{max value upper camber} - \text{min value lower camber}) * (y/c_{\text{max}} - y/c_{\text{min}})$$

$$C_m = (ac - x/c_{\text{max}}) * (C_l * \cos(AoA) + C_d * \sin(AoA))$$

100 rpm

$$C_l = (1.7) (18/21 - 3/21) = 1.207$$

$$C_d = (1.7) (1.5/21 - 0.5/21) = 0.085$$

200 rpm

$$C_l = (3) (18/21 - 3/21) = 2.13$$

$$C_d = (3) (1.5/21 - 0.5/21) = 0.15$$

300 rpm

$$C_l = (-0.7) (18/21 - 3/21) = -0.49$$

$$C_d = (-0.7) (1.5/21 - 0.5/21) = -0.035$$

Table 6.3. Results of C_l , C_d at 100RPM, 200RPM and 300RPM

100 RPM		200 RPM		300 RPM	
C_l	C_d	C_l	C_d	C_l	C_d
1.207	0.085	2.13	0.15	-0.49	-0.035

CHAPTER 7

SUMMARY AND CONCLUSION

Finally, after conducting the in-depth investigation using CFD-based approaches, to understand the various impacts of both the shape and layout of the vortex generators on the power efficiency of the large wing turbines, it can be concluded that: CFD analysis is used to know the impact of the VG design parameters which showed that the VG reduced the occurrence of separation flow and drag, while increasing the lift at high angles of attack. VGs installed blade analysis flow characteristics proved that the VG reduced the area of separation by generating tip vortices thus letting the air with higher kinetic energy to flow to the area of comparatively low energy. These research results are useful for selecting the VG's shape and its position to mounting by performing a full-scale simulation by automating the construction of the simulation environment. The selected VGs (spikes) discussed here are presently installed in a 2.3 MW wind turbine and are being observed using SCADA systems. The VGs can also be installed on other large wind turbines by using the suggested method for creating a computational environment. Large reduction in the power generating performance of a wind turbine is due to the surface roughness, which can be avoided to some part, by using Vortex generator arrays in between 14%-41% of the blade span that increases AEP by 1%.

REFERENCES

- [1] Introduction to Wind Turbine Aerodynamics by A. P. Schaffarczyk
- [2] Calvert N.G, "Wind Power Principles", Charles Griffin & Co London, 1979.
- [3] Grasso, F. Development of Thick Airfoils for Wind Turbines. In Proceedings of the 50th AIAA Aerospace Sciences Meeting, Nashville, TN, USA, 9–12 January 2012; p. AIAA2012-0236.
- [4] Kishinami, K.; Taniguchi, H.; Suzuki, J.; Ibano, H.; Kazunou, T.; Turuhami, M. Theoretical and experimental study on the aerodynamic characteristics of a horizontal axis wind turbine. *Energy* 2005, 30, 2089–2100.
- [5] M. B. BRAGG AND G. M. GREGOREK (1987). Experimental Study of Airfoil Performance with Vortex Generators.
- [6] Qing Tian, Dave Corson and Jonathan Baker (2016). Application of Vortex generators to Wind Turbine Blades.
- [7] Hans Mueller-Vahl, Georgios Pechlivanoglou, C.N. Nayeri, C.O. Paschereit (2012). Vortex Generators for Wind Turbine Blades: A combined Wind Tunnel and Wind Turbine Parametric study.
- [8] Hyeon-Gi Moon, Sunho Park, Kwangtae Ha and Jae-Ho Jeong (2021). CFD-Based In-Depth Investigation of the Effects of the Shape and Layout of a Vortex Generator on the Aerodynamic Performance of a Multi-MW Wind Turbine.
- [9] Martinez, J.; Bernabini, L.; Probst, O.; Rodriguez, C. An Improved BEM Model for the Power Curve Prediction of Stall-regulated Wind Turbines. *Wind Energy* 2005, 8, 385–402.
- [10] Jeong, J.; Yoo, C.; Lee, J.; Kim, K.H.; Choi, J.-W. 3-Dimensional Flow Field Analysis and Tip Shape Design in a Wind Turbine Blade. In Proceedings of the Korean Society for Computational Fluids Engineering, Spring Conference, Jeju, Korea, 26 May 2011; pp. 243–248.
- [11] Wu, Z.; Chen, T.; Wang, H.; Shi, H.; Li, M. Investigate aerodynamic performance of wind turbine blades with vortex generators at the transition area. *Wind Eng.* 2021, 1–15, 0309524X211038542.
- [12] Øye, S. The effect of vortex generators on the performance of the ELKRAFT 1000 KW turbine. In Proceedings of the Aerodynamics of Wind Turbines: 9th IEA Symposium, Stockholm, Sweden, 11–12 December 1995; pp. 9–14.
- [13] Hansen, M.O.L.; Velte, C.M.; Øye, S.; Hansen, R.; Sørensen, N.N.; Madsen, J.; Mikkelsen, R. Aerodynamically shaped vortex generators. *Wind Energy* 2016, 19, 563–567.