International Journal of Engineering Research and Development
ISSN: 2278-067X, Volume 1, Issue 2 (May 2012), PP.18-24
www.ijerd.com

Optimum Aerodynamic Design in Wind Mill Blades using Winglet function

Prof, G Sivaraj[1], Mr. M Gokul raj[2]
[1]*Department of Aeronautical Engineering, Bannari Amman institute of technology, Sathyamangalam, India*
[2]*Department of Aeronautical department, Bannari Amman Institute of technology, Sathyamangalam, India.*

***Abstract*—**Technology has the enemy of nature in one way .But sometimes technologies do come out as an exception to the above rule. Technological advancements have improvised them over time. In this paper we shall glance at the features. Wind energy is the most popular renewable energy resource thanks to its elastic cost compared with conventional fossil resources. In order to increase the use of wind energy, it is important to develop wind turbine rotor models with high rotation rates and power coefficients. This study aimed at manufacturing highly efficient wind turbine rotor models using NACA profiles.

***Keyword*—**Technology, Wind turbine rotors, Wind Energy, Turbine Rotor, NACA profile.

I. INTRODUCTION

Wind energy is the most popular renewable energy resource thanks to its elastic cost compared with conventional fossil resources. In line with advancing technology, manufacturing costs are expected to fall below the current level. In Europe, wind energy capacity level for2010 is 40,000MW and that level is ten times higher than that of 2000. Turkey also needs to increase the use of wind energy in order to raise its electricity production capacity to 60GW for 2010.Like Turkey, some countries have clean energy resource blade rying no fossil based energy reserves. However, fossil energy resources such as blade on dioxide pollute the world and they are a threat to future generations. Therefore, utilizing wind energy potential is crucial in global world context. In order to increase the use of wind energy, it is important to develop wind turbine rotor models with high rotation rates and power coefficients. This study aimed at manufacturing highly efficient wind turbine rotor models using NACA profiles. The term ''rotor'' refers to blades and rotor as a whole.

II. MODEL DIMENSIONS

Scale model of this project is NTK/41 WIND TURBINE.

i. Scale ratio 1:120
ii. Model dimensions

Rotor Diameter	: 340 mm
Hub Diameter	: 60 mm
Blade length	: 140 mm
No. of blades	: 3

A. *wind turbine dimension 41/TK*

M41	: Rotor diameter
M35	: Hub diameter
Blade length	: 16.8 m
No. of blades	: 3

As the technology developed, wind turbines were also improved just like the other technologies until recent times and they are continuously evolving. As it was mentioned earlier there are also VAWTs which geometrically differ from HAWTs.

The world's largest horizontal axis wind turbine built on Hawaii Island, manufactured by Boeing Aerospace Industry. This turbine has a rotor diameter of 97.5m and has a rotor swept area of 7,470m^2. Its rated power is 3.2 MW [2].

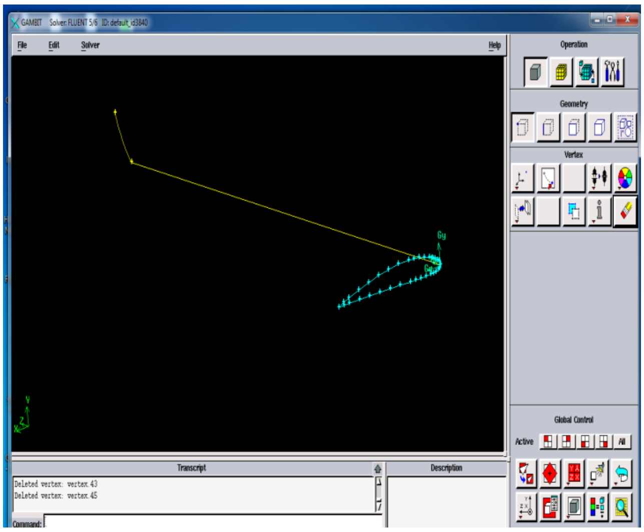
HAWTs are most preferable wind power machines due to their effectiveness when compared with VAWTs, but VAWTs have some superiorities upon HAWTs. One of them is they do not need any yaw mechanism and their installation is more easy and so their maintenance as well.

B. WINGLET FUNCTION

The art is then to design a winglet, which optimizes drag reduction, maximizes power production and minimizes thrust increase. The resulting pressure difference on an operating wind turbine blade causes inward span wise flow on the suction side and outward span wise flow on the pressure side near the tip. At the trailing edge, vorticity is generated, which is the origin of induced drag. A winglet is a load carrying device that reduces the span wise flow, diffuses and moves the tip vortex away from the rotor plane reducing the downwash and thereby the induced drag on the blade

The main purpose of adding a winglet to a wind turbine rotor is to decrease the total drag from the blades and thereby increase the aerodynamic efficiency of the turbine. Reduction of total drag is obtained if the additional drag from the winglet is less than the reduction of the induced drag on the remaining blade.

The model was designed as per the literature survey taken from journal. Next this model was analyzed in computational software. Then the results are compared with the literature survey. The dimensions are taken from the NTK/41 wind turbine.



```
Deleted vertex: vertex.43
Deleted vertex: vertex.45
```

Fig. 1 figure denotes the winglet sketch

C. MESHING

An inflated boundary of prismatic elements was used near the blade surface to improve spatial resolution and gain a better understanding of boundary layer phenomena. An unstructured mesh with polyhedral elements was used for volume meshing. Simulations were carried out with the turbulence model, coupled with a blend factor of 0.5 for the advection scheme.

The computational mesh was constructed automatically using polyhedral cells mesh, surrounded at solid boundaries by three prismatic extrusion layers. Because polyhedral cells fill space more efficiently than tetrahedral elements, fewer cells were required than might otherwise have been needed, significantly aiding the goal of using a small desktop machine to perform such aerodynamic analyses.

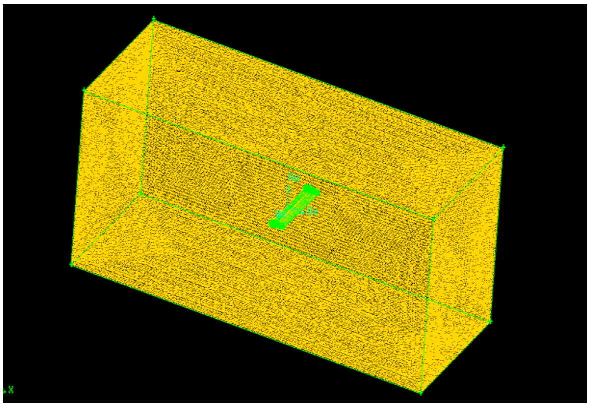


Fig. 2 3D view of final polyhedral mesh with volume source visible around the blade

A model that has already been meshed and it has only 130’000 polyhedral cells. Note that at least 5 million cells, with hexagonal in the near-wall regions, would be necessary to obtain reliable and detailed results in such a case.

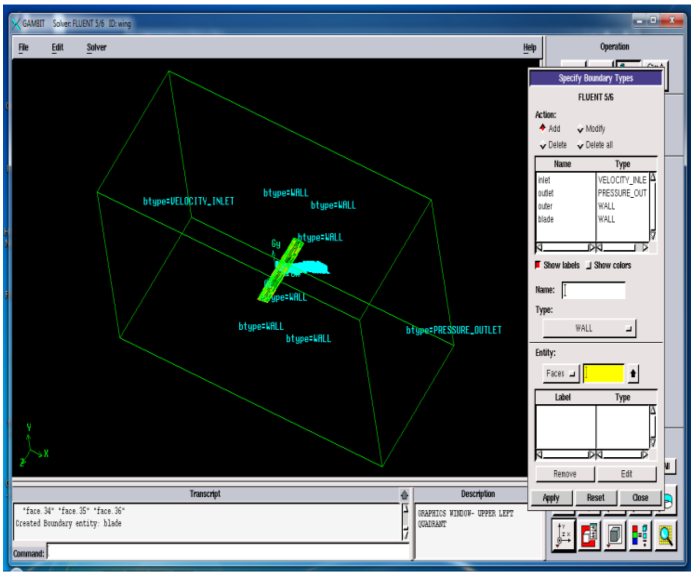
The computational domain extends far upstream of the blade where the boundary condition will be a velocity inlet. The top and bottom of the computational domain are "periodic" boundary conditions, which mean that whatever flows out of the top goes directly into the bottom. It is assumed that since the outer limits of the computational domain are so far apart, this blade behaves as if in an infinite free stream. In order to adequately resolve the boundary layer along the blade wall, grid points will be clustered near the wall. Far away from walls, where the flow does not have large velocity gradients, the grid points can be very far apart. A hybrid grid will be used in this problem. Grid adaptation within the flow solver, Fluent, will increase the grid density even more near the wall and wherever else needed.

D. DEFINE THE GEOMETRY

When the geometry was defined in the creation of the computational mesh, all faces of the domain were assigned names. The names of the inlet and outlet planes (at $x = 0$ and $x = L$) are front face and back face of domain as velocity inlet and pressure outlet respectively. The names of the planes at $y = L$, $z=0$, *and* $z=L$ are outer wall as wall. The names of the model are blade as a wall. And bottom face is defined as road.

Dimension of The Domain

- Height = 15m
- Length = 35 m
- Breath = 20m



Name	Type
inlet	VELOCITY_INLE
outlet	PRESSURE_OUT
outer	WALL
blade	WALL

"face.34" "face.35" "face.36"
Created Boundary entity: blade

GRAPHICS WINDOW- UPPER LEFT QUADRANT

Fig. 3 defining the geometry

E. IMPORTING THE MESH

Fluent reads the grid with about 130’000 cells from gambit file. Grid Check is sure there is no negative volume or face area and there is no warning of any kind.

F. DEFINE THE PHYSICAL MODEL

Define the model of Solver is Segregated for Continuity equation is first solved for all cells, then Momentum and then turbulences. This works well for incompressible and moderate compressible flow. Applying the Implicit for each equation is solved for all cells together with actual dates. The implicit solver brings faster convergence.

Define the model as 3D and Steady (blade velocity will be constant and we don’t Expect instabilities). It is Absolute there is no moving mesh zone in the mesh. Define the Model is Viscous as k-epsilon for a robust and efficient turbulent model which gives good results in most cases where turbulences have an isotropic repartition. Define the model of energy equation.

G. SPECIFY MATERIAL PROPERTIES

Define the materials is air And it is properties of

Density =	1.225 kg·m-3
Viscosity=	1.464e-5 kg·m-1·s-1

Those values correspond to the ICAO norm. Fluent means dynamic viscosity as we consider air as incompressible and are not looking for heat transfer problematic, we don’t need to specify properties.

H. BOUNDARY CONDITIONS

Let the 101325 Pa which corresponds to the ICAO-Norm. Fluent.Blade model is "wall" with "blade" (in the field "Zone Name").We considers our model as a wind-tunnel model. So the blade is a stationary wall, the viscosity makes the air stick at the blade coachwork, so no slip the coachwork is very smooth, so a roughness of zero. Ceiling of the wind-tunnel and Side wall of the wind-tunnel are specified shear for this will allow the air to slip on the ceiling wall. This is not realistic, but so, we can use a very coarse mesh without boundary layer problems. Velocity is 5 m·s-1 in the Speed field. Correspond to 90km/h. and 0.05m in the "Roughness Height" field.

I. PROCESSING

Initialize the "Compute From"-inlet. This will attribute to all cells of the model, the velocity, pressure and turbulences values that we defined for the inlet. Calculate a solution for using corresponding selected condition.Normally we would have to enable better numerical schemes (2nd or 3rd order and run until a much better convergence of the flow solution is reached, but this would take about 3 hours with this case and about 2 weeks with an adequate mesh refinement). So we simply visualize the actual results.

J. FLOW PARAMETERS TAKEN FOR STUDY

Seven important results were obtained from the analysis

1. drag force variation along the blade model
2. lift force variation along the blade model
3. Static pressure variation along the blade model
4. Total pressure variation along the blade model
5. Velocity vectors
6. Path line of velocity magnitude variation along the blade model

III. RESULTS AND ANALYSIS

The Meshed geometry is analyzed using the given Boundary conditions specified for FLUENT and the variation of flow parameters are plotted and studied.

CONTOUR PLOTS (FILLED) FOR MODEL: Pressure coefficient variation of wing

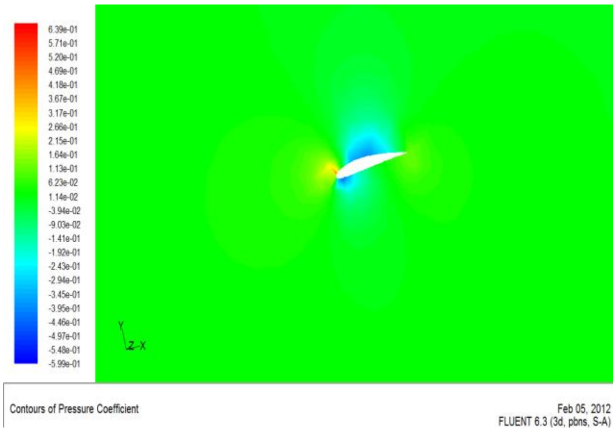
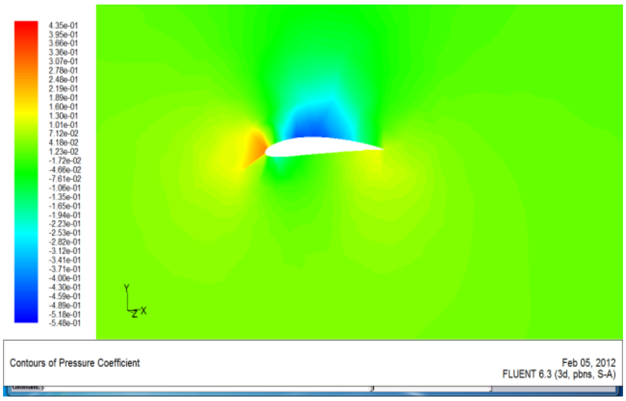


Fig. 4 Pressure coefficient variation of wing

A. Pressure coefficient variation of wing with winglet

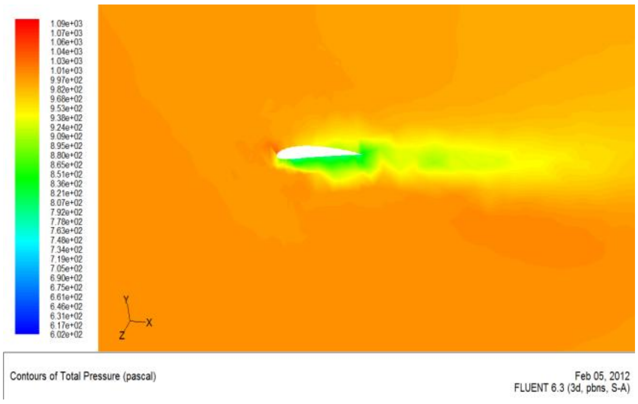


Contours of Pressure Coefficient

Feb 05, 2012
FLUENT 6.3 (3d, pbns, S-A)

Fig. 5 Pressure coefficient variation of wing with winglet

B. Total Pressure variation of wing

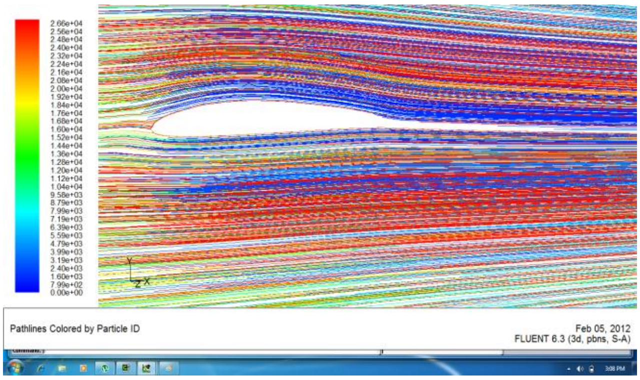


Contours of Total Pressure (pascal)

Feb 05, 2012
FLUENT 6.3 (3d, pbns, S-A)

Fig. 6 Total Pressure variation of wing

C. Path line of model Pressure coefficient variation of wing



Pathlines Colored by Particle ID

Feb 05, 2012
FLUENT 6.3 (3d, pbns, S-A)

***Fig. 6* Path line of model** Pressure coefficient variation of wing

TABLE 1: RESULT OF WING TURBINE WITH OUT MODIFICATION

Force vector: (1 0 0) -→ Drag force
Force vector: (0 1 0) -→ Lift force

Wing without modification	**Drag force**	**Lift force**
Pressure force N	194.79107	425.26551
Viscous force N	34.021567	0.89831118
Total pressure N	228.81263	426.16382

TABLE 2: RESULT OF WING TURBINE WITH WINDMILL

Force vector: (1 0 0) -→ Drag force
Force vector: (0 1 0) -→ Lift force

Wing with winglet	**Drag force**	**Lift force**
Pressure force N	136.91423	1411.6327
Viscous force N	32.167544	-0.12091241
Total pressure N	169.08178	1411.5118

IV. CONCLUSION

There is no doubt that adding a winglet to the existing blade can change the downwash distribution leading to increased produced power, but a load analysis must be made whether the additional thrust can be afforded.

Finally, the effect of pointing the winglet towards the suction side (downstream) was investigated. Based on the above mentioned results the twist distribution form *winglet* was used and resulted in a slightly improved winglet compared to *winglet*. But still there is the issue of tower clearance.

For comparison a rectangular modification of the original blade tip was designed with the same platform area as the blades with winglets. This modification does produce more power compared to the original blade but not as much as the cambered and twisted winglets. All four upwind pointing winglets do result in lower thrust compared the rectangular blade tip, while the downwind pointing winglet results in comparable or even higher thrust.

Based on the present investigation it is seen that *winglet* has the best overall power performance of the upwind pointing winglets, but the increase in power of around 1.3% for wind speeds larger than 6 m/s is relatively low and must be compared to the increase in thrust of around 1.6%. But pointing the winglet downstream seems to increase the power production even further. The effect of sweep and cant angles is not accounted for in the present investigation and could improve the performance of the winglets even more.

V. ACKNOWLEDGMENT

We first thank our ‘GOD’, the supreme power for giving us a good knowledge and our parents for making us study in a renowned college We owe a great many thanks to my colleagues and friends for their help and encouragement.

REFERENCES

[1]. Manwell J. F. et.al "Wind Energy Explained", John Wiley & Sons Ltd., England, 2002

[2]. Spera, D. "Wind Turbine Technology", ASME Press, 1994

[3]. Buhl, M. L., Jr. "WT_PERF User's Guide", NREL, Colorado, 2004

[4]. Scheck, S. "HAWT Aerodynamics and Models from Wind Tunnel Measurements, NREL, Colorado, 2002

[5]. Glauert, H. "The Elements of Aerofoil and Airscrew Theory", University Press, Cambrige, 1959

[6]. Bertin, J. J. and Smith, M. L. "Aerodynamics for Engineers", Third Edition, Prentice Hall, New Jersey, 1998

[7]. Katz J, Plotkin J. "Low Speed Aerodynamics", McGraw- Hill Inc., New York, 1991

[8]. Jureczko.M, Pawlak.M, Mezyk.A Optimisation of wind turbine blades, journal of materials processing technology 167(2005).

[9]. Sohn.Y.U, Chun.H, Kim.Y.C, Chung.C.W, Kim.Y.H, Han.K.S. Blade design of a 750Kw direct- drive wind turbine generator system.

[10]. Kong C, Kim H, Kim J. A study on structural and aerodynamic design of composite blade for large scale HAWT system. Final report, Hankuk Fiber Ltd; 2000.

[11]. Numerical Analysis of Winglets on Wind Turbine Blades using CFD Jeppe Johansen† and Niels N. Sørensen†,‡ †Wind Energy Department, Risø National Laboratory, Technical University of Denmark

[12]. Wind rotor blade construction Small Wind Systems for Battery Charging Contract R 7105 By Teodoro Sanchez Campos ITDG, Sunith Fernando and Hugh Piggott1