Finite element analysis of Horizontal Axis Wind Turbine blade by ANSYS

- **Vu Cong Hoa**
- **Nguyen Huu Tien**

Department of Enginneering Mechanics, Ho Chi Minh city University of Technology, VNU-HCM
(Manuscript Received on 30th Oct., 2015, Manuscript Revised 10th Nov., 2015)

ABSTRACT

The wind turbine blade is a very important part of the rotor. Extraction of energy from wind depends on the design of blade. In this paper, the authors based on the blade element theory (BET) and the model of an optimum rotor developed by Glauert's to design a 1000-mm-long horizontal axis turbine blade model. This model was then used for the finite element analysis. The authors also used the code of the commercial finite element ANSYS to conduct analyses. The results from the linear static structural analysis revealed that the best design provides adequate stiffness and strength to produce the proposed power without any structural failure.

Key words: *Design, turbine blade, wind turbine, linear static, FEA, ANSYS.*

1. INTRODUCTION

Wind energy is very well advertised and has been added to many grids. Much has been learned from the advances in wind turbine blade design, but many differences exist that must be addressed. Also, most wind turbine blades are hollow to reduce self-weight and cost, but this type of design is not practical for an ocean current turbine blade. If water were to leak into the hollow region, via a crack, the system would become negatively buoyant and sink, and could possibly damage other units in the array. Wind turbines [2], [3], [5], [6], [10], are subjected to very specific loads and stresses. Due to the nature of wind, loads are highly variable. Varying loads are more difficult to handle than static loads because the material becomes fatigued. Moreover as a working medium the air is of low density so that the surface required for capturing energy must be large. The change of the shape of blade [10], [11], [12], is one of the methods to modify stiffness and stability, but it may influence aerodynamic efficiency of wind turbine. Other method to change dynamic and mechanical properties of wind turbine is modifying the composite material, which the blade is made of. A global nonlinear FE model of the entire blade was prepared and the boundaries to a more detailed sub-model were extracted. The FE model was calibrated based on full scale test measurements. The model is based on an extreme value analysis of the load response process in conjunction with a stochastic representation of the governing tensile strength of the rotor blade material [1], [4], [6], [12].
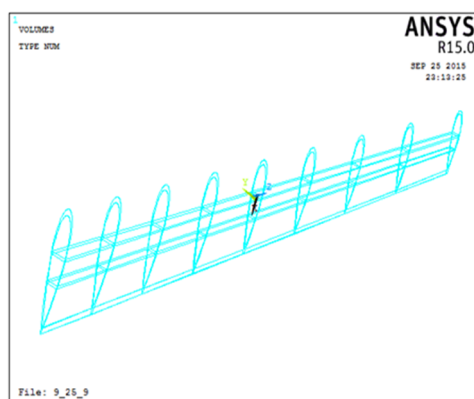
2. MODEL

The CATIA software was used to design the blade under profiles NACA 0012
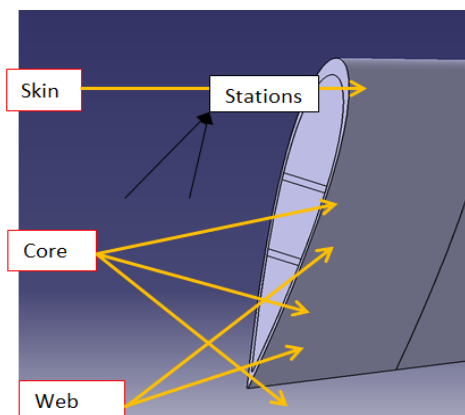
Figure 1. All component of blade

After design in CATIA, the model was imported into ANSYS APDL.

Figure 2. Geometry in ANSYS APDL

This model is divided into three parts which are Skin, Web, Core, with completely different materials. The parameters of the model is based on NACA 0012.

For Skin and Web materials used are: S2Glass / SP125S, element types used inANSYS APDL is SOLSH190 For Core, materials used is Steel code AS4 / 35 016, the element type used in ANSYS APDL is SOLID 186.

Figure 3. 2-D cross-section of recommended design

3. METHOD ANALYSIS

The finite element method (FEM) is very useful and has traditionally been used in the development of wind turbine blades for investigating the global behavior in terms of, for example, Eigen frequencies, tip deflections, and global stress/strain levels.

A big advantage of using FEM is that, once the model is set up and calibrated, complex load cases representing actual wind conditions can be analyzed. Only idealized loads can be imposed in a full scale test and in this paper the critical flap wise load case is evaluated. The FE model of the wind turbine blade is created using APDL language in ANSYS [14], [18]. Then the results will be reevaluated, verify by ANSYS WORKBENCH.

4. FINITE ELEMENT MODELING USING ANSYS

All models were generated using the bottom-up solid modeling method in ANSYS Classic. The skin and web material properties were entered as orthotropic and the stacking sequence was defined using the section data command. The skin and webs were meshed using SOLSH190 [15], a linear layered 3-D, 8-node, 3 degree of freedom (DOF) per node element. The isotropic core material was meshed with SOLID186 [15], a 3-D 20-node, 3 DOF per node quadratic element [15]. By meshing in the order of linear to quadratic all mid side nodes are eliminated resulting in proper element connectivity. The EORIENT command was used to ensure that the skin and web elements were properly aligned. Fig. 4 is a meshing that is used

for section 1 of of designs. A linear static structural analysis [15] and an eigenvalue buckling analysis [17] were conducted for each of the designs. The flap wise pressure distribution was applied and the root was fixed in degrees of freedom (Fig. 5).
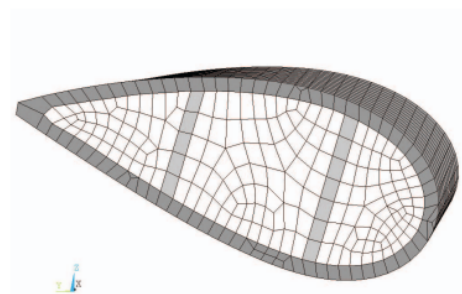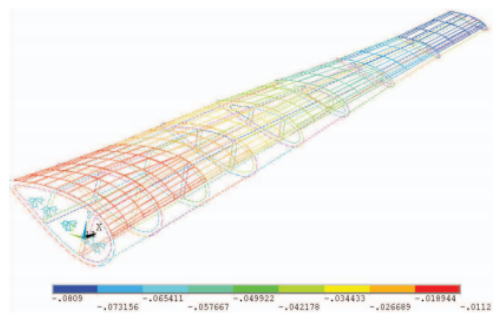

Figure 4. Mesh used for section 1 of designs

Figure 5. Flap wise pressure distribution and boundary conditions applied to FE model; units in MPa.

Static Analysis, Loading, and Boundary Condition

The linear static analysis is specified using the ANTYPE, STATIC command [18]. The nodes at the root are constrained in all directions by coupling the root areas using the DA command. These steps can be found in the appended code. The flap wise pressure distribution is applied to the low pressure side of the blade as a surface load. The pressures are transferred by default from the geometry to the

FE model, where they are applied to face 6 on each skin element. The negative pressures act out of the elements. Table 1 and Fig. 6 show the material properties of skin, Web and Core of blade are used in ANSYS APDL.

Table 1. Material properties used for blade analyses

| Properties		AS4/3501-6	S2-Glass/XP125S	HCP100
Material system		Prepreg	Prepreg	
Fiber type		Graphite	S2-Glass	
Resin type		Epoxy	Epoxy	
Fiber volume		60%	60%	
Composite density	g/cm³	1.58	1.98	0.4
Lamina elastic properties				
E_1	GPa	143	51	~0.700
E_2	GPa	10	17	
E_3	GPa	10	17	
G_{12}	GPa	6	7	~0.210
G_{23}	GPa	3	7	
G_{13}	GPa	5	7	
v_{12}		0.3	0.25	~.45
v_{23}		0.52	0.32	
v_{13}		0.3	0.25	
Lamina strength properties				
F_{1t}	MPa	2172	1779	
F_{1c}	MPa	-1558	-641	
F_{2t}	MPa	54	58	
F_{2c}	MPa	-186	-186	
F_{3t}	MPa	59	58	13.5
F_{3c}	MPa	-186	-186	-10.5
F_4	MPa	87	75	7.3
F_5	MPa	94	77	
F_6	MPa	124	77	
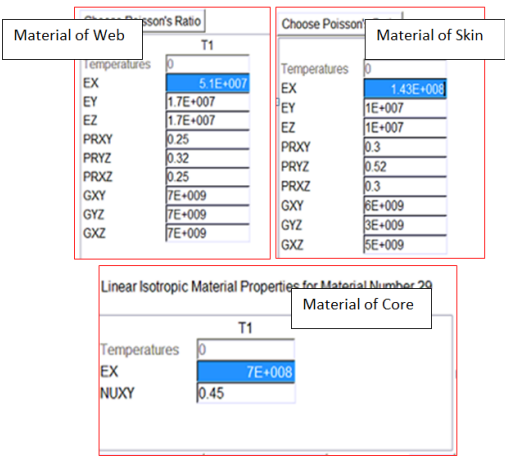
Figure 6. Material Properties of Skin, Web, Core in ANSYS APDL

In this paper the SOLSH190 [11] was used for layered applications such as modeling

laminated shells or sandwich construction. The layered section definition is given by section (SECxxx) commands. Accuracy in modeling composite shells is governed by the first-order shear-deformation theory (also known as Mindlin-Reissner shell theory). SOLSH190 is the most well suited element for modeling the skin and webs since it was designed for simulating shell structures with a wide range of thickness. SOLID186 [11] was used for the core material.

Fig. 7 and Fig. 8 show the meshing blade and grid meshing at area. Fig 9 and 10 show the properties of layer of skin and web, respectivity.

Fig. 11 shows a boundary condition of fixed support at first section of turbine blade. Table 2 is the code for siloving in ANSYS APDL.
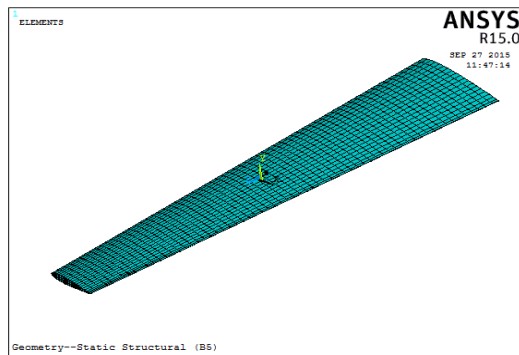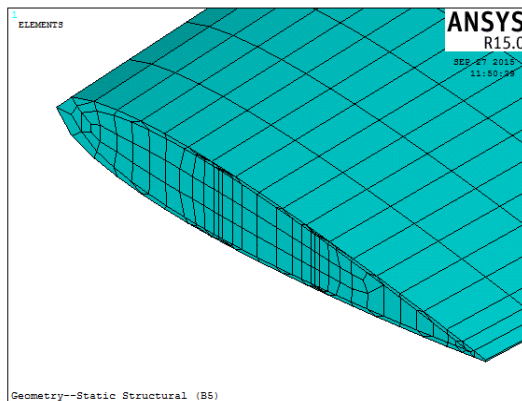

Figure 7. Meshing blade

Figure 8. Grid meshing at area

Figure 9. Property of layer of skin

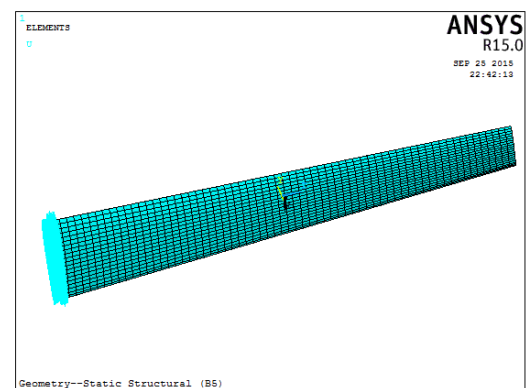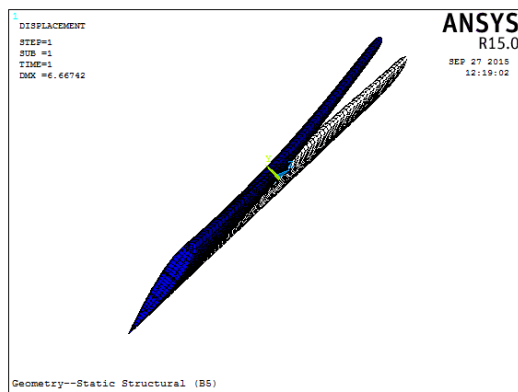**Figure 10.** Property of layer of web

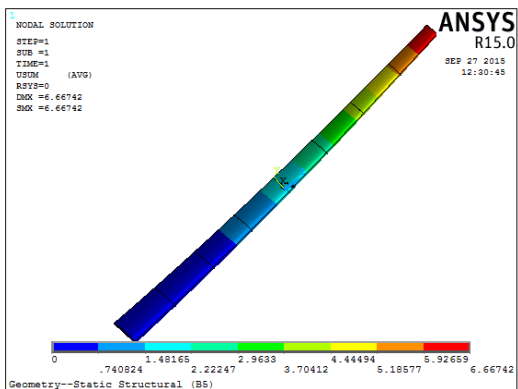**Figure 11.** Boundary conditions of fixed support at first section

Table 2. Code for solving in ANSYS APDL

```
/UNITS,MKS !Units are in
m, MPa, Newton, kg, and
kg/m3
/prep7
ANTYPE,STATIC
ET,1,190
ET,2,186
ET,3,190
!Material properties for the
skin and web were taken
from
!Material properties for: uni
AS4/3501-6          orthotropic
laminate-Mat 1
uimp,1,ex,ey,ez,143E3,10E
3,10E3
uimp,1,gxy,gyz,gxz,6E3,3E
3,5E3
uimp,1,prxy,pryz,prxz,0.3,0.
52,0.3
mp,dens,1,1580
!Material     prop     for:
Divinycell HCP 100-Mat 2
(really ortho,v~.67)
!input    as    isotropic    ~
assumption
mp,ex,2,700
mp,prxy,2,.45 !assummption
mp,dens,2,400
!Material properties for: uni
S2-glass/XP251s orthotropic
laminate-Mat 3
uimp,3,ex,ey,ez,51E3,17E3,
17E3
uimp,3,gxy,gyz,gxz,7E3,7E
3,7E3
uimp,3,prxy,pryz,prxz,0.25,
0.32,0.25
mp,dens,3,1980  !CF  skin
layup
SECTYPE,1,SHELL
!16layers  making  16  mm
skin
SECDATA,.001,1,-45
!default to 3 int pts
SECDATA,.001,1,45
SECDATA,.001,1,0
```

```
SECDATA,.001,1,0
SECDATA,.001,1,0
SECDATA,.001,1,0
SECDATA,.001,1,0
SECDATA,.001,1,0
SECDATA,.001,1,0
SECDATA,.001,1,0
SECDATA,.001,1,0
SECDATA,.001,1,0
SECDATA,.001,1,0
SECDATA,.001,1,45
SECDATA,.001,1,-45
keyopt,1,6,0
keyopt,1,8,1
keyopt,1,10,0
!S2 web layup
SECTYPE,3,SHELL
!16 layers making 16 mm
web
SECDATA,.001,3,-45
!default to 3 int pts
SECDATA,.001,3,45
SECDATA,.001,3,-45
SECDATA,.001,3,45
SECDATA,.001,3,-45
SECDATA,.001,3,45
SECDATA,.001,3,-45
SECDATA,.001,3,45
SECDATA,.001,3,-45
SECDATA,.001,3,45
SECDATA,.001,3,-45
SECDATA,.001,3,45
SECDATA,.001,3,-45
SECDATA,.001,3,45
SECDATA,.001,3,-45
SECDATA,.001,3,45
keyopt,3,6,0
```

```
keyopt,3,8,1
keyopt,3,10,0
FINISH !Exit pre-procesor
module
/SOLU !Solution module
ANTYPE,STATIC        !Set
Static Analysis
!Apply Boundary Condition
and Pressure on top
DA,1,ALL,0
DA,2,ALL,0
DA,3,ALL,0
DA,4,ALL,0
DA,5,ALL,0
DA,6,ALL,0
SFA,70,,PRESS,-.0112
SFA,86,,PRESS,-.0165
SFA,101,,PRESS,-.0237
SFA,117,,PRESS,-.03186
SFA,133,,PRESS,-.0398
SFA,149,,PRESS,-.052
SFA,165,,PRESS,-.0678
SFA,180,,PRESS,-.0809
SOLVE !Solve current load
state
FINISH      !Exit   solution
module
!Failure Criteriea
!CF Skin
FC,1,s,xten,2172
FC,1,s,xcmp,-1558
FC,1,s,yten,57
FC,1,s,ycmp,-186
FC,1,s,zten,59
FC,1,s,zcmp,-186
FC,1,s,xy,87
FC,1,s,yz,94
FC,1,s,xz,124
FC,1,s,XYZP,-1
FC,1,s,XZCP,-1
FC,1,s,XZCP,-1
```

```
!Foam
FC,2,s,xten,13.5
FC,2,s,xcmp,-10.5
FC,2,s,yten,13.5
FC,2,s,ycmp,-10.5
FC,2,s,zten,13.5
FC,2,s,zcmp,-10.5
FC,2,s,xy,7.3
FC,2,s,yz,7.3
FC,2,s,xz,7.3
!S2 Webs
FC,3,s,xten,1779
FC,3,s,xcmp,-641
FC,3,s,yten,58
FC,3,s,ycmp,-186
FC,3,s,zten,58
FC,3,s,zcmp,-186
FC,3,s,xy,75
FC,3,s,yz,77
FC,3,s,xz,77
FC,3,s,XYCP,-1
FC,3,s,YZCP,-1
FC,3,s,XZCP,-1
!-----------------------------
!For Eigenvalue Buckling
Analysis enter the
following, but after the
static
!Analysis has been solved
/SOLU
ANTYPE,BUCKLE
BUCOPT,LANB,1
MXPAND,1
SOLVE
FINSH
/POST1
SET,FIRST
PLDISP,1
FINSH
```
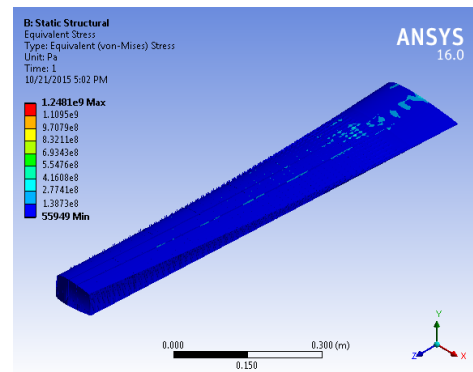
5. RESULTS

(a)

(b)

Figure 12. Deformation of blade (a) in APDL (b) in ANSYS workbench

Figure 12 (a) and (b) show the deformation of blade. The maximum deformation of blade is 6.6742 mm, according to standard IEC 61400-1 is a small and safe [20]. The results obtained in ANSYS Classic (APDL) and ANSYS workbench exactly alike.

Figure 13 shows the results of von mises stress. The maximum stress is 1248.1 MPa, this compared with the maximum stress of the material AS4 / 3501-6 is 2137 MPa (310 KSI). Blade eligibility durable [19].

Figure 13. The result of von- Mises Stress

6. CONCLUSIONS

In this paper, the ANSYS software was used to analyse finite element of blade. By using ANSYS Workbench and ANSYS APDL to check the accuracy of the result, the authors found that the result is nearly completely accurate with a minor difference. The results show that the best design provides adequate stiffness and strength to produce the proposed power without any structural failure. Conducting the finite element analysis FEA for the wind turbine's wings is very important as it help to get high-performance wing profiles. Applying the FEA analysis in Ansys ADPL is more difficult than that in Ansys Workbench.

After the analysis, the results were compared with the standards of design and wing calculations.

The results are durable and meet the criteria within the permitted level of international standards.

The results are completely identical when using either Ansys Workbench or Ansys APDL for analyzing.

Phân tích phần tử hữu hạn lưỡi Turbine gió trục ngang bằng ANSYS

- **Vũ Công Hòa**
- **Nguyễn Hữu Tiến**

Bộ môn Cơ kỹ thuật, Trường Đại học Bách khoa, ĐHQG-HCM

TÓM TẮT

Các lưỡi tuabin gió là một phần rất quan trọng của các cánh quạt. Khai thác năng lượng từ gió phụ thuộc vào thiết kế của lưỡi. Trong bài báo này, các tác giả dựa vào lý thuyết phần tử cánh (Blade Element Theory – BET) và mô hình rotor tối ưu được phát triển bởi Glauert để thiết kế một lưỡi tuabin trục ngang dài 1000 mm. Sau đó mô hình này được sử dụng cho các phân tích phần tử *hữu hạn. Ngoài ra, các tác giả sử dụng code của chương trình phần tử hữu hạn thương mại ANSYS để tiến hành các phân tích. Các quan sát thấy rằng, việc sử dụng công cụ số để thiết kế lưỡi cắt cùng với việc phân tích kết cấu tĩnh tuyến tính chỉ ra rằng thiết kế tốt nhất đảm bảo đủ độ cứng và độ bền để tạo ra năng lượng đề xuất mà không có bất kỳ hư hỏng cấu trúc.*

Từ khóa: Thiết kế, lưỡi tuabin, tuabin gió, tĩnh tuyến tính, FEA, ANSYS.

REFERENCES

[1]. Brøndsted P., Lilholt, H., Lystrup, Aa. *Composite materials for wind power turbine blades.* Materials Research Department. Risø National Laboratory, (2005).

[2]. DNV and Riso National Laboratory, *Guidelines for Design of Wind Turbines*, Second Edition, Printed by Jydsk Centraltrykkeri, Denmark 2002, ISBN 87-550-2870-5.

[3]. Hau. E., Wind Turbines Fundamental, Technologies, Application, Economics. Krailling, Springer, (2006).

[4]. Berggreen, C., Branner, K., Jensen, J. F., Schultz, J. P. *Application and Analysis of Sandwich Elements in the Primary Structure of Large Wind Turbine Blades,* Sandwich Structures and Materials Volume 9, November, (2007).

[5]. Martin O.L Hansen. Aerodynamics of Wind Turbines. 2nd ed, London and Virginia: Earth scan, (2008).

[6]. Isaac M. Daniel, Ori Ishai. Engineering Mechanics of Composite Materials. 2nd ed. New York: Oxford University Press, Inc. (1996).

[7]. Mahmood M. Shokrieh, Roham Rafiee, *Simulation of fatigue failure in a full composite wind turbine blade*, Composite Structures, Volume **74**, Issue 3, 332-342, August (2006).

[8]. ANSYS, Inc. Release 15.0, Elements Reference. Part 1. Element Library, (2015).

[9]. Jensen F.M., Falzon B.G., Ankersen J., Stang H., Structural testing and numerical simulation of a 34m composite wind turbine blade, Composite structure 76, 52 – 61, (2006).

[10]. One Dimensional Variations: Blades, Dutch Offshore Wind Energy Converter Project, LM Glasfiber Holland BV, (2003).

[11]. Ronold Kunt O., Larsen Gunner. *Reliability, Based design of wind turbine rotor blades against failure in ultimate loading*, Engineering Structures (22), 565 – 574 (2000).

[12]. Edwards K.L., Davenport C. *Materials for rotationally dynamic components: rationale for higher performance rotor-blade design*, Materials & Design, Volume **27**, Issue 1, 31-35 (2006).

[13]. Edward C. Smith, Zhang, Jianhua, *Structural Design and Analysis of Composite Blade for a Low Weight Rotor.*

[14]. ANSYS® Academic Research, Release 16.0, Help System*, Mechanical APDL Theory Reference*, ANSYS, Inc

[15]. ANSYS® Academic Research, Release 16.0, Help System, *Structural Guide,* ANSYS, Inc.

[16]. ANSYS® Academic Research, Release 16.0, Help System, *Composites*, ANSYS, Inc.

[17]. ANSYS® Academic Research, Release 15.0, Help System, *Linear Buckling Guide*, ANSYS, Inc.

[18]. ANSYS® Academic Research, Release 15.0, Help System, *Commands Reference,* ANSYS, Inc.

[19]. 3501-6 Epoxy Matrix High Strength, Damage-Resistant, Structural Epoxy Matrix, Hexcel Corporation, 281 Tresser Boulevard, 16th Floor, Stamford, CT 06901-3261, USA.

[20]. EUOPEAN STANDARD, Wind turbine generator systems, Part1: Safety requirements, IEC 61400-1:999, English version, February (2004).