Published by :
http://www.ijert.org
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 10, October-2019

Aeroelastic Analysis of Rotating Blade using Two-Ways Fluid-Structure Interaction

Chandra Prakash
M. Tech, Department of Mechanical Engineering,
PMCE, Sonipat, India

Rahul Malik
HOD, Department of Mechanical Engineering,
PMCE, Sonipat, India

Pappala Ramakrishna
Assistant Professor,
Department of Mechanical Engineering,
PMCE, Sonipat, India

Abstract - **This study deals with fluid-structure interaction (FSI) which is one of the emerging fields of numerical calculation and simulation. FSI concerns have a really major role to play in several engineering and scientific fields, yet a comprehensive study of such concerns remains a challenge due to their multidisciplinary and strong non-linear behavior. FSI happens when a fluid encounters with a solid structure, it creates pressure on it that can lead to structure deformation. The deformed shape affects the flow field in return. The modified moving fluid imposes a different pattern of stresses on the structure, which are repetitive in nature.**

In this report, The FSI study was conducted on a rotating blade of the wind turbine. The study has been conducted using the partition method in which two ways of couplings were used to simulate the fluid-structural interactions.The appropriate measurements of the fluid and structural model are created using ANSYS APDL. Transient structural and CFD fluent is used as a pre-processing tool to model the entire computational domain and volume mesh. ANSYS Mechanical (transient structural) is used for structural to evaluate the dynamic response of a structure under unsteady fluid pressure loads. In ANSYS Workbench, The two solvers (ANSYS Fluent & Mechanical) are coupled (exchange of data) using System Coupling. A modal analysis has performed to evaluate a structure member's dynamic response.

Key Words: *FSI Interaction, Aerodynamics, Ansys Mechanical, critical frequency, deformation and*

1. INTRODUCTION

Aero elasticity is the analysis of interactions between the elastic, inertial and aerodynamic forces acting on an elastic structure that is exposed to fluid flow. In the classical elasticity theory, the deformation is usually assumed to be minimal and the change in dimensions is often ignored and calculations are made on the basis of the original shape. Moreover, the situation is different in the case of aero elasticity. In this case, elastic deformation plays an important role in calculation of external force as deformation is also included in the calculation. Aero elasticity phenomena play a major role in blade development.

1.1 Flutter: Flutter is one the significant problem in aero-elasticity which leads to large amplitude vibration ends up to the structural failure.Flutter happens due to the interference of two structural modes: plunge and rotational. An airfoil with two basic degrees of freedom or natural modes of vibration: plunge and pitch.

Even though, for flutter to occur on the structures there are many critical conditions, two of them are listed below:

- The insufficient difference in frequency of first torsional mode and flap wise bending mode.
- The aerodynamic centre of the blade cross-section is placed before the mass of the blade

1.2 Fluid-Structure Interaction (FSI)

Fluid-Structure Interaction (FSI) is multi-physics coupling which deals with the interaction of deformable or movable structure with the surrounding fluid (fig.1). It can be stable or oscillatory .While designing many engineering structures e.g. wind turbine blade, aircraft wing, FSI is a crucial factor. Failure to consider the effects of oscillatory interactions can be catastrophic, especially when structures that include fatigue-prone materials. One of the most notorious examples of large scale failure is probably Tacoma Narrows Bridge (1940).

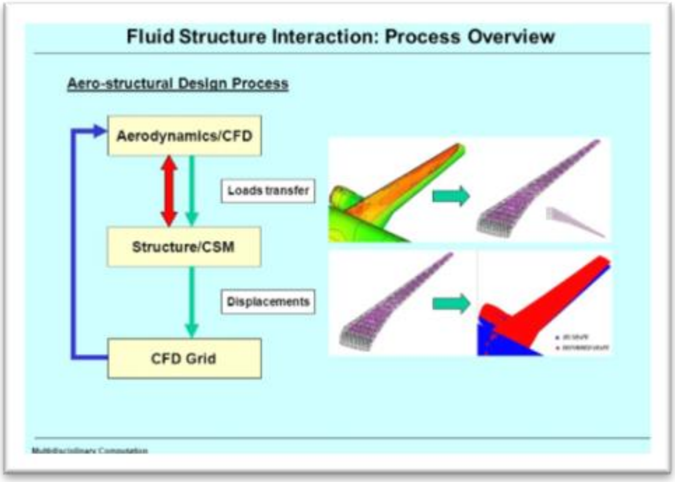


Fig.1:FSI: Process overview

2. BLADE GEOMETRY

The geometry of wind turbine blade is generated by using ANSYS APDL script [8]. The baseline blade structure includes a leading edge panel from 0-15% chord, a box spar from 15%-50% chord, and trailing edge panel from 50%-100% chord. The box spar is made with two shear webs at 15% and 50% chord.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 10, October-2019

The box spar begins at 25% r/R and tapered off to zero at the blade tip. Length of the blade is 33.25m.

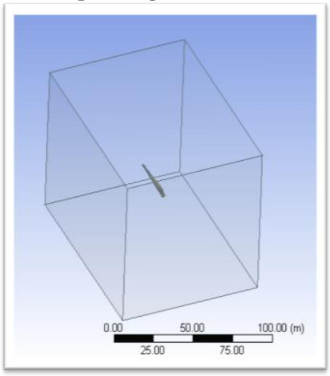


Fig.2: Structural member inside fluid domain.

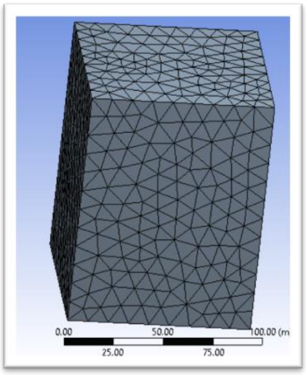


Fig.3: Fluid domain mesh

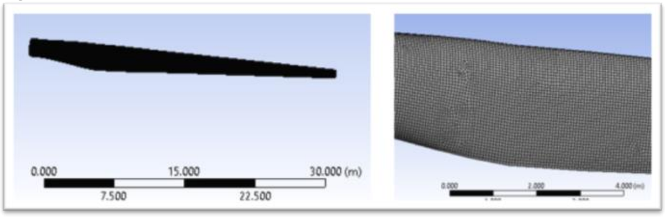


Fig.4: Structural mesh

The fluid domain is constructed as a rectangular prism where blade is mounted. To prevent the velocity gradients at the upper boundary and reflections from the outlet boundary respectively, the vertical and horizontal length of the domain is taken as approximately 50 m. Fig.2 shows the blade's closer view within the fluid domain. For the CFD mesh, a triangular mesh is used to create the surface mesh, Then the volume mesh is generated. Volume mesh is created by tetrahedral mesh that assigned to the unstructured mesh group. 20 Inflation layers maintained near the surface of the blade to provide better mesh close to the boundary. Unlike structured mesh, it is not possible to identify unstructured mesh cells using i, j, k index. Fig.3 shows the fluid domain mesh.

2.1 Fluid domain mesh

For the finite element mesh, ANSYS APDL script is used [8]. Shell element is used to create the surface mesh. The mesh size used in this model is 0.2 m. Fig.4 shows the structural mesh.

3.0 MATERIAL PROPERTIES

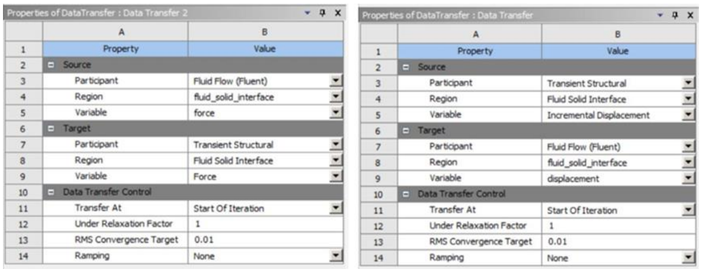
Table.1 shows the properties of the material used to formulate the blade.

Property	A260 Uniaxial Fabric	CDB340 Triaxial Fabric	Spar Cap Mixture (70% uni & 30% triax)	Random Mat	Balsa	Gel Coat	Fill Epoxy
Ex (GPa)	31	24.2	27.1	9.65	2.07	3.44	2.76
Ey (GPa)	7.59	8.97	8.35	965	2.07	3.44	2.76
Gxy(GPa)	3.52	4.97	4.7	3.86	0.14	1.38	1.1
vxy	0.31	0.39	0.37	0.3	0.22	0.3	0.3
vf	0.4	0.4	0.4	-	N/A	N/A	N/A
wf	0.61	0.61	0.61	-	N/A	N/A	N/A
Density (g/cm^3)	1.7	1.7	1.7	1.67	0.144	1.23	1.15

Table 1: Material Properties summary.

Data Transfer

Data transfer is used to transfer the data between the ANSYS Fluent and Transient structure. Fig.5 shows the detailed view of data transfer.



Properties of DataTransfer : Data Transfer 2

	A	B
1	Property	Value
2	Source	
3	Participant	Fluid Flow (Fluent)
4	Region	fluid_solid_interface
5	Variable	force
6	Target	
7	Participant	Transient Structural
8	Region	Fluid Solid Interface
9	Variable	Force
10	Data Transfer Control	
11	Transfer At	Start Of Iteration
12	Under Relaxation Factor	1
13	RMS Convergence Target	0.01
14	Ramping	None

Properties of DataTransfer : Data Transfer

	A	B
1	Property	Value
2	Source	
3	Participant	Transient Structural
4	Region	Fluid Solid Interface
5	Variable	Incremental Displacement
6	Target	
7	Participant	Fluid Flow (Fluent)
8	Region	fluid_solid_interface
9	Variable	displacement
10	Data Transfer Control	
11	Transfer At	Start Of Iteration
12	Under Relaxation Factor	1
13	RMS Convergence Target	0.01
14	Ramping	None

Fig.5: Properties of Data transfer

Results: Modal analysis of the blade is performed to validate the result from the reference report

Mode Number	Frequency	
	Reference	Outcome
1	1.0783	1.0505
2	1.7001	1.699
3	29804	2.9309
4	5.0382	4.9613
5	6.093	6.393
6	10.305	10.013

5.2 TOTAL DEFORMATION AND EQUIVALENT STRESS OF BLADE

Fig.6 and Fig.7 show the total deformation and equivalent stress of the blade. Inlet velocity for this condition is 12m/s and blade is rotating with 43.2 rpm. Damping factor considers for this case is 0.1. Duration of the analysis is 10s.

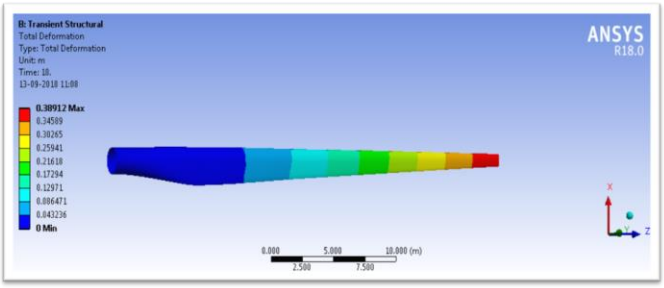


Fig.6: Total deformation contour.

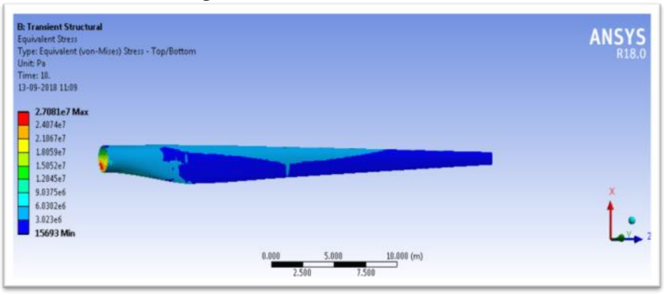


Fig.7: Equivalent Stress contour.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 10, October-2019

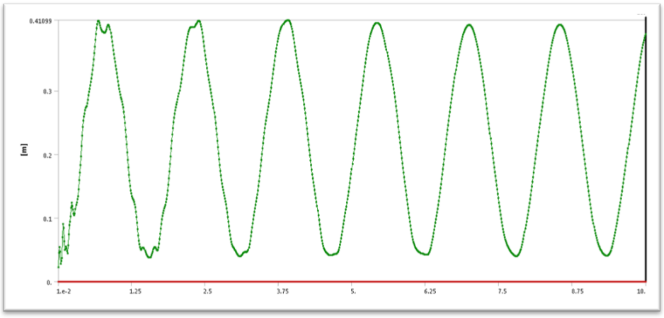


Fig.8: Total deformation vs time curve

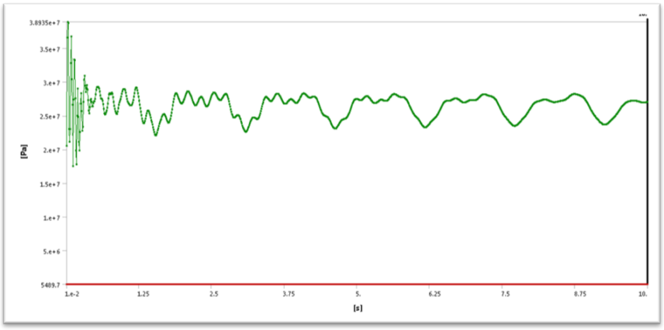


Fig.9: Equivalent stress vs time curve

Fig.8 and Fig.9 show the total deformation and equivalent stress of the blade. Inlet velocity for the next condition is 14 m/s and blade is rotating with 43.2 rpm. Damping factor considers for this case is 0.02. Duration of the analysis is 5s.

CONCLUSION:

By performing the two ways fluid structure interaction on the wind turbine blade, we have concluded the following points:

- The total deformation contours shown in fig.6 and fig.8 indicates that maximum value of deflection occurs at the blade tip. The total deformation value increases gradually from the root to tip of the wing.
- The equivalent stress contours shown in fig.7 and fig.9 indicates that maximum value of stress occurs at the blade root i.e. 26.7 MPa.
- For a given velocity there is an aerodynamic load and because of that load, stress induced on the structure. The given velocity is below the flutter velocity so that aeroelastic damping damp out the oscillatory motion.

REFERENCES:

[1] Lin Wang, R. Quant, A. Kolios “Fluid structure interaction modelling of horizontal-axis wind turbine,” Journal of Wind Engineering and Industrial Aerodynamics, pp. 11-25, 2016.
[2] ANSYS Workbench 17.0 -user guide.
[3] E. H. Dowell, H. C. C., “A Modern course in Aeroelasticity,” ISBN 0-7923-0062-9, Second revised and enlarged edition, 1989.
[4] Roham Rafiee, M. Tahani, M. Moradi “Simulation of aeroelastic behavior in acomposite wind turbine blade,” Journal of Wind Engineering and Industrial Aerodynamics, pp. 60-69, 2016.
[5] P. Elangovan, “Fluttering analysis in wind turbine blade,” Sweden, 2012.
[6] S. Zheng, L.P. Chua, Y. Zhao “Simulations of Fluid-Structure Interaction of Wind Turbine,” 2016.
[7] S. El Mouhsine, K. Oukassou, M.M. Ichenial, B. Kharbouch, A. Hajraoui “Aerodynamic and structural analysis of wind turbine blade,” in 11th International Conference Interdisciplinarity in Engineering, Romania, 2017.
[8] [8] B.R. Resor, T. Bushnell, “A 1.5 MW NuMAD Blade Model,” Sandia National Laboratories, California.