DESIGN OPTIMIZATION OF KAPLAN TURBINE

Muhammad Mahmood Aslam Bhutta [1], Ahmad Naveed [1], Muhammad Aftab Ahmad [2], Syed Amjad Ahmad [3], Amanat Ali Bhatti [4], Ali Akbar [5]

[1] Department of Mechanical Engineering, University of Engineering & Technology, Lahore, Pakistan.
[2] School of Engineering & Technology, ICBS, Lahore, Pakistan.
[3] Faculty of Mechanical Engineering, NFC Institute of Engineering and Fertilizer Research, Faisalabad, Pakistan.
[4] Department of Petroleum Engineering, University of Engineering & Technology, Lahore, Pakistan.
[5] Department of Mechanical Engineering, Rachna College of Engineering &Technology, UET, Lahore, Pakistan.
*Corresponding Author: bhuttathermo@yahoo.com

ABSTRACT: *In this work Finite Element Analysis Technique is used to analyze and optimize the blades of a Kaplan Turbine. Instead of using a customized software for turbine design calculation, 3D solid modelling and analysis. General purpose software Microsoft Excel, Pro. E and ANSYS are used. The optimized FEA model is validated by applying the same technique on Kaplan turbine installed at Nandipur Power Station, Gujranwala. This work is an effort to predict the stress concentrations and crack initiation points by using FEA techniques. Comparison of results is made with the actual damaged blade of the turbine installed at Nidipur Hydal Power plant. A design modification is proposed for the blade. The analysis after design modification showed that the new design has less stress concentration area hence better operational life is predicted.*

Keywords: Finite Element Analysis, Kaplan Turbine, Design Optimization

INTRODUCTION

Hydraulic turbines are present in most countries and accounts today to about one fifth of the total electricity production in the world [1]. Furthermore, they are producing electricity with high efficiencies (up to 95%) with zero pollution [2].

Pakistan is one of the developing economies of the world. Pakistan's energy demand is growing all the time. When discussing the potential energy options for Pakistan the hydel energy certainly look like the best one, because we have the largest canal system in our country. For installing the new hydro projects on the canals and rivers we need modern and simple techniques for Kaplan turbines designing and analysis. Since the Kaplan Turbines are considered to be the one of the best choice when water head is low [3].An important component of a medium and low head reaction turbine (Kaplan Turbine) is the main rotor of the turbine. Because it is responsible for creating torque for rotation of the generator rotor [4]. It subjected to a large proportion of stresses due to hydraulic pressure in the system [5] However, it is one of the most challenging parts to design, due to the interaction of many complex flow features, such as unsteadiness, turbulence, separation, streamline curvature, secondary flow, swirl, and vortex breakdown [6].

KAPLAN TURBINE

The Kaplan Turbine is a pure reaction type hydraulic turbine. They work efficiently at low head and high discharge. Following are the operating conditions for a Kaplan turbine:

Head = 2-30 m

Flow rate = 70-800 m^3/s

With proper adjustment of blades during its run, the Kaplan Turbine is capable of giving high efficiency for a wide range of load conditions. The pitch of runner blades is automatically adjusted by the governor through the action of a servomotor [7]. Figure 1 shows a simplified diagram of a Hydraulic Turbine and related components:

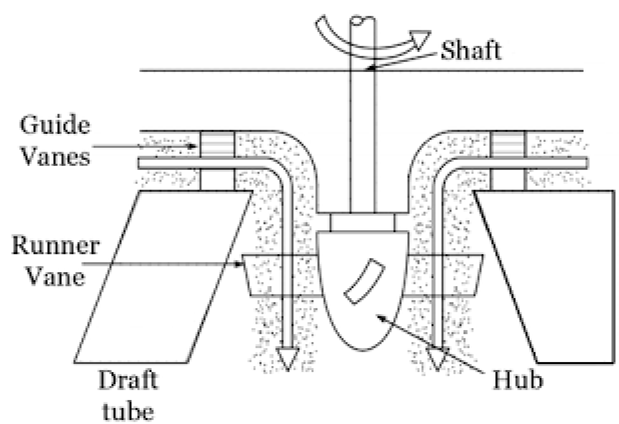

Figure 1: Simplified Diagram of Energy of Hydraulic Turbine and Related Components

DESIGN OF THE RUNNER OF KAPLAN TURBINE

A numerical study comparable to the reference [8] Design of Kaplan turbine was performed on MS Excel based work sheets. The design parameters were

Net Head = 6.71 m (22 ft)
Discharge = 86.03 m^3/s (3040 cusecs)

Based on these parameters the calculations performed using MS Excel worksheets is shown in the following Tables.

Table 1: Power Calculation

Power	Head	H	6.707317073	m
	Discharge	D	86.032	m^3/sec
	Speed Ratio	v	2	2.0-2.1
	Flow Ratio	f	0.67	0.63-0.7
	Specific Weight	w	9810	N/m^3
	Efficiency	η	0.81	
	Power	P	4.6	MW

Table No.2 Outer Diameter of the Runner Calculation

Outer Diameter Of Runner	Outer Diameter of Runner	D_o	4	
	Hub or Boss Diameter	D_b	0	
	Hub to Outer Dia Ratio	D_b/D_o	0.42	0.35-0.61
	Flow Velocity	V_f	7.685970669	m/sec
	Peripheral Velocity	u	22.94319603	m/sec
		D_o	4.159848734	m
		D_o	13.64430385	ft

Table No.3 Hub Dia, Speed and Torque Calculation

Hub Diameter	Diameter of Hub	D_b	1.747136468	m
	Diameter of Hub	D_b	5.730607616	ft
Speed	Speed in rpm	N	105.3362261	rpm
Torque	Torque	T	415677.4925	Nm

Table No 4: Runner Vane Angles Calculation

Runner Vane Angles	Analysis at Hub Section				
	Peripheral Velocity at Hub	u_1	9.636142331	m/sec	
	Hydraulic efficiency	η_h	0.95		
		V_{u1}	6.486915543	m/sec	
	Runner Vane angle at inlet	β_1	112.2807291	deg	
	Runner Vane angle at outlet	β_2	38.57652956	deg	
	Analysis at Extreme Edge of Runner				
	Peripheral Velocity at Extreme	u_1	22.94319603	m/sec	
		V_{u1}	2.724504528	m/sec	
	Runner Vane Angle at inlet	β_1	159.1861077	deg	
	Runner Vane Angle at Outlet	β_2	18.52084948	deg	

Table No 5: Shaft Diameter Calculation

Shaft Diameter	Assumptions				
	Factor of Saftey	N	2		
	Material	Low Carbon Cold Rolled Steel SAE 1020			
	Ultimate Tensile Strength	S_{ut}	65000	psi	
	Yield Strength	S_y	38000	psi	
	Notch Radius	r	0.2	in	
	Geometric Stress Conc. Factor	K_{ts}	2		
	Calculations				
	Torque	T	3679036.783	in-lb	
	Notch Sensitivity	q	0.86		
	Fatigue Stress Concentration	K_{fs}	1.86		
	Shaft Diameter	d	373.4006934	mm	

MODELING OF TURBINE

Pro-Engineer wildfire-5 was used for CAD modeling. All parts were made as part model according to the drawings. These parts were imported in the assembly module for complete assembly of the turbine [9].

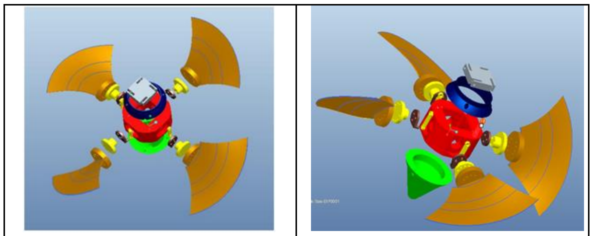

Figure 2: Exploded Views of the Runner Solid Model in Pro-E

The assembly was checked for any overlap of the components and it was corrected accordingly before analysis. Figure 2 shows an exploded view of assembly of Kaplan Turbine [10].

SIMULATION

Simulation was done in ANSYS WORKBENCH V-12 [11]. The 3D model of the blade was exported to ANSYS Design Modular. SETP file format was found to be the best for import process and then loads and supports were applied on it. The static analysis was performed using FEA approach. Three different forces act on the blade during its working. First is due to its mass (FM) that will act downward. Second is centrifugal force (Fc). Third is water force (Fw) [12],[13]. There will be two components of the water force one is tangential (Fwt) and other is axial (Fwa).

A. Assumptions

Following assumptions were made for primary analysis of turbine.

- Turbine is running at full load
- Turbine is running at maximum discharge
- Turbine is running at maximum head

These assumptions were used throughout the analysis.

B. Boundary Conditions

Available power $P = \rho\, g\, Q\, H$

$H = 22$ feet $= 6.7$ m

$Q = 3040$ Cusecs $= 86.082\ m^3/s$

$g = 9.81\ m/s^2$

$\rho = 1000\ kg/m^3$

Mass of the blade =1620 kg

Rated speed = 107 RPM

Diameter of the rotor = 4.3688 m

C. Centrifugal Force Calculations

$Fc = mr\omega^2$

$Fc = (1620 \times 2.19 \times ((2 \times 3.14 \times 107)/60)^2)$

$Fc = 444938$ N

D. Water Force on the Blade

Force due to water = $F_w = \rho\, Q\, V$

$F_w = \rho\, Q\, (r\, \omega)$

$F_w = 1000 \times 86.082 \times 2.19 \times ((2 \times 3.14 \times 107)/60)$

$F_w = 2111244$ N

No. of blades = 4

Water force on a blade = 2111244/4 = 527811 N

Approximate angle of the blade is 10^o then the axil and tangential components of the water force are given by:

Tangential component of water force:

$F_{wt} = 527811 \times (\cos 10^o)$

$F_{wt} = 519792$ N

Axial component of water force:

$F_{wa} = 527811 \times (\sin 10^o)$

$F_{wa} = 91653$ N

Table No. 6 summarizes the different load acting on the blade.

Table 6: Net applied Loads

Sr. No.	Force Type	N
1	Weight	15890
2	Centrifugal Force	444938
3	Water Force	527811
4	Water Force Axial	91653
5	Water Force Tangential	519792

ANALYSIS AND RESULTS IN WORK BENCH

After applying the loads according to above Table 1, the simulation was performed. Statics of mesh are given in Table. 5.

During static structural analysis VON MISES equivalent stress were calculated and stress contours were displayed shown in Fig. 11. Other than this contours of Maximum Principal Stress and Maximum Principal Elastic Strain were also displayed [14]. (can be seen in Fig 12 and Fig. 13).

Table 7: Statistics of mesh

Bodies	1
Active Bodies	1
Nodes	75066
Elements	41393
Mesh Metric	Element Quality
Minimum	1.0405 E-02
Max	0.9994
Average	0.7372
Standard Deviation	0.1527

These figures shows that maximum stresses are at the junction points of the blade and connecting hub. As this is the same location from where the blade of Kaplan Turbine unit No. 1 at Nandipur Power Plant was broken (see Fig. 6).

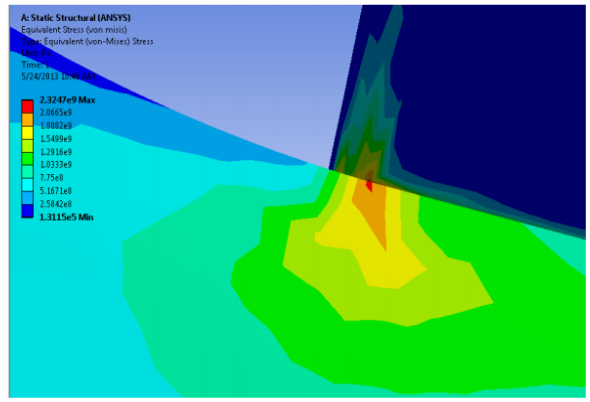

Figure3: Von Mises Stresses near the Stress Concentration

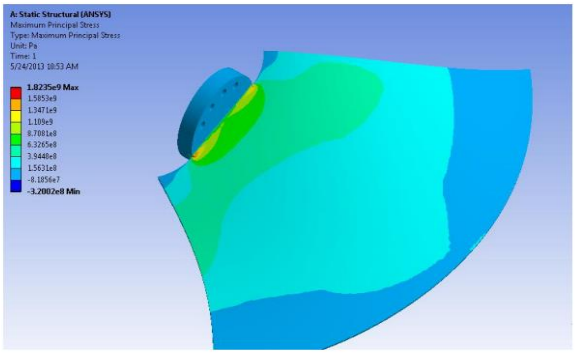

Figure 4: Maximum Principal Stress

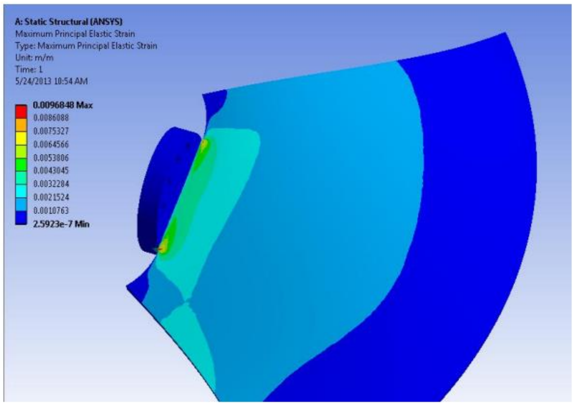

Figure 5 : Maximum Principal Elastic Strain

Figure 6: Photograph of Actual Broken Blade

DESIGN MODIFICATION

The blade design was modified by creating stress relief groove at the maximum stress concentration point. The groove is of 1 inch diameter was made on the both sides of the blade. After making the groove the same loads are applied and the analysis was carried out to observe the effects of that stress relief groove.

Figure 7 a, 7 b and 7 c show the von Mises stresses, after design modification.

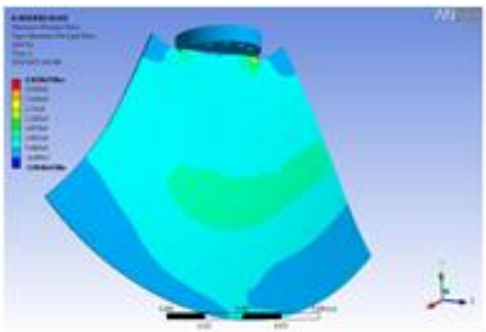

Figure 7 a: Von Mises Stresses after Design Modification

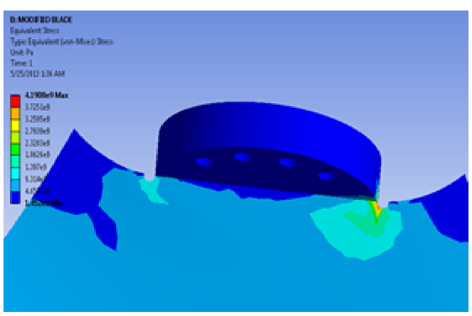

Figure 7 b: Von Mises Stresses after Design Modification (close up)

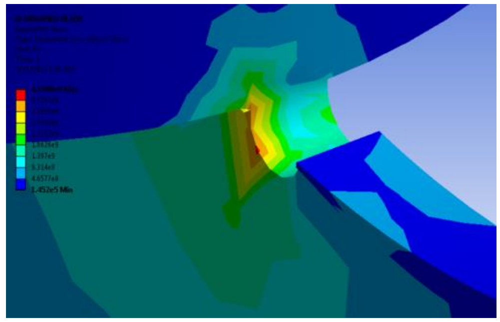

Figure 7 c: Von Mises Stresses after Design Modification (close up)

And the Figure 8 and Figure 9 shows the Maximum Principal Stresses and Maximum Principal Strains, respectively. These figures reinforce the finding in the previous results.

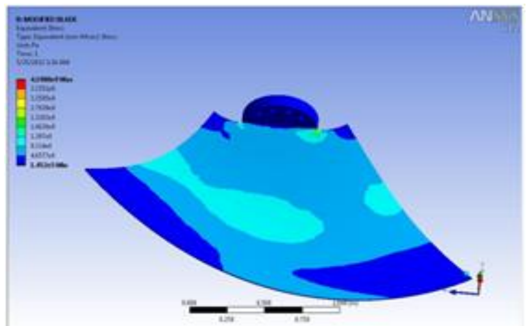

Figure 8: Maximum Principal Stresses after Design Modification

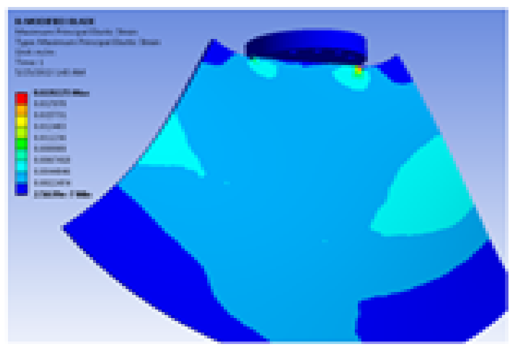

Figure 9: Maximum Principal Strain after Design Modification

CONCLUSION

This work is an effort to predict the stress concentrations and crack initiation points in the turbine rotor blades by using FEA techniques. Simulation is used for the prediction of cracks. From the physical inspection of the broken blade it is clear that the blade is broken at the same point from where the simulation predicted. The design modification was recommended; blade was redesigned and tested with computer simulation again. From the results it was concluded that new design has less stress concentration areas and longer life can be expected.

REFERENCES

[1] K.V. Alexander, E.P. Giddens and A.M.Y. Fuller: "Axial-Flow Turbine For Low Head Microhydro System'', Renewable Energy, Volume 34, Issue 1, January 2009, Pages 35–47.

[2] Loiseau F "Importance of Draft Tube In Rehabilitation Projects", IAHR 24thSymposium on Hydraulic Machinery and Systems, Foz do Iguazzu, Brazil 2008.

[3] Skotak, A., Obrovsky, J., "Analysis Of The Flow In The Water Turbine Draft Tube In Fluent And CFX", 25th CADFEM Users' Meeting , Dresden , Germany,2007.

[4] Lukas M., Ales S., Jiri ., 2010, "Conditions of Kaplan Turbine Cfd Analysis" Ansys Conference 2010.

[5] A. Arnone, M. Marconcini, F. Rubechini, A. Schneider "Kaplan Turbine Performance Prediction Using CFD: An Artificial Neural Network Approach" HYDRO 2009 Conference paper n.263, Lyon, France, 26-28 October 2009.

[6] K. Bennett, J. Swiderski and J. Huang, "Application of CFD Turbine Design for Small Hydro Elliott Falls, A Case Study" Waterpower XII, Salt lake City, 2001.

[7] B. D. Marjavaara "Automatic Shape Optimisation of A Hydropower Draft Tube" 4th asme_jsme Joint Fluids Engineering Conference, Honolulu, Hawaii, USA, July 6–11, 2003.

[8] R. Khare and V. Prasad "Numerical Study on Performance Characteristics of Draft Tube of Mixed Flow Hydraulic Turbine" Hydro Nepal, Jan 2012.

[9] G.T. Csanady, Theory of Turbo machines, McGraw Hill, New York, 1964.

[10] B. D. Marjavaara "CFD Driven Optimization of Hydraulic Turbine Draft Tubes using Surrogate Models" Ph D thesis, Department of Applied Physics and Mechanical Engineering, Lulea University of Technology; 2006.

[11] ANSYS Workbench 14.0 User Guide

[12] B.D. Marjavaara , T.S. Lundström, "Response Surface-based Shape Optimization of a Francis Draft Tube"2005.

[13] B. D. Marjavaara, and T. S. Lundstr: "Redesign Of A Sharp Heel Draft Tube By A Validated CFD-Optimization" 2005.

[14] M. K. Shukla, R. Jain, V. Parsad, S.N. Shukla "CFD Analysis of 3D Flow for Frances Turbine"Aug 2011.

.