FEM Analysis of Turgo Impulse Turbine Blade

Sourabh KHURANA[1,*], Varun GOEL[2] and Anoop KUMAR[2]

[1]School of Engineering and Technology, Shoolini University, Bajhol, Solan 173212, India
[2]Department of Mechanical Engineering, National Institute of Technology, Hamirpur 177005, India

([*]Corresponding author's e-mail: sourabhnith@gmail.com)

Abstract

The present research work describes the development of Turgo turbine blades on the Solidworks software. Finite element simulation (Ansys V14) has been used for analysis of stress and total deformation produced inside the Turgo impulse turbine. Finite element simulation is effective when it is used to analyze the strain and stress distribution. It has been observed during analysis that the maximum stress occurs at the root of blade suction side.

Keywords: Ansys, turbine blades, stress, deformation

Introduction

Energy is required in each field of commercial and human activities. Energy generation is one of the key factors for economic and social development in both the developed and developing nations of the world. The different sources of energy are hydro, thermal, nuclear and non-convectional energy resources like wind, solar and biomass. Presently in India the total installed capacity including all the resources is 182344.62 MW, of which the share for hydro energy is 21.22 % [1]. Among the different sources of energy, hydropower is recognized as a renewable source of energy, which is economical, non-polluting and environment friendly. Water has been used as an energy source for thousands of years, with ancient civilizations using water to drive mills through the use of water wheels. Turgo is an impulse turbine which can be used for medium to high head applications. They have flat efficiency curves and provide exellent part load efficiencies, so they can be used for large flow rate variations and thus allow for efficient operation in lower head ranges [2]. The Turgo turbine runner resembles a pelton runner split in half at the plane of symmetry. Turgo turbines are designed while Pelton and Francis head ranges overlap. They can be used in head ranges from 15 to 300 m [3]. In Turgo turbines, the water jet enters from one side of the runner and exits from the other. In the present paper, analysis of stresses and deformation inside the blade of Turgo turbine has been studied using Ansys software. The blades of the Turgo turbine were simulated with Solidworks software (Version 2009).

Cobb and Sharp [2] constructed laboratory scale test fixture to test the operating performance characteristics of impulse turbines. Tests were carried out to determine the effect of turbine efficiency of variations in speed ratio and jet misalignment on two Turgo turbines and then results were compared with a Pelton turbine. They have found that at a speed ratio of 0.46 Turgo turbine efficiency was over 80 %. Tests on jet misalignment showed that moving the jet to the inside and outside edge of the turbine blades caused a drop in Turgo efficiency of 10 - 20 % and optimal speed ratio by 6.5 %.

Perrig [4] has conducted a series of experimental tests on Pelton turbine models and analyzed the flow. Under numerical simulation they presented wall measurments based on a two phase homogenous model. The five distinct zones were defined by results obtained and the results were compared by numerical simulation. The flow pattern in the bucket was analyzed from the results.

Thapa *et al.* [5] developed a computer program to create and optimize design of Francis

runners. They exported their final design to computational fluid dynamics (CFD) and fluid structure interaction (FSI) for analysis. They have concluded that the runner outlet diameter, peripheral velocity at inlet and blade angle distribution have the highest effect on sediment erosion of Francis runners. Indeed, the blade profile alone reduces sediment erosion by more than 30 %.

Neopane *et al*. [6] has conducted sediment erosion analysis on Francis turbine used at CAHUA power plant by using Ansys CFX software. The author studied the effect of sediment concentration and size on the erosion of the Francis turbine. The author concluded that maximum erosion was found to be on the pressure side of the blade outlet and at the shroud. They have even observed sediment erosion at the suction side of guide vane. They have also concluded that if the particle size in the water increases beyond a critical particle size, the turbine should not be operated at low guide vane opening.

Nakanishi *et al*. [7] has observed the flow simulation in a Pelton turbine bucket by using moving particle semi-implicit method (MPS) simulation. A numerical code has been developed and this code has been validated by experiment. The free surface flow depends upon the position of the bucket and pressure distribution on the bucket computed by numerical code and this result compared well with experiment.

Vesely and Varner [8] have conducted model testing to upgrade a 62.5 MW Pelton turbine. CFD software was used for flow simulations and finite element stress analysis was also conducted for the runner and nozzle. From their study, they concluded that the rate capacity increased by about 6 %, power increased by 9 % and efficiency increased by 1.4 %.

Koukouvinis *et al*. [9] used the smoothed particle hydrodynamics (SPH) method to compare the performance of two different types of Turgo runners in comparison to fluent runners. They concluded that the SPH method gave the same results produced by the CFD solver but in a much reduced time.

Khare *et al*. [10] used Ansys CFX software to analyze 3D turbulent real flow in a Francis turbine, for three guide vane opening and different rotation speed. In the study, they analyzed the average values of flow parameters including velocities and flow angles at the runner inlets and outlets, guide vanes and stay vanes of the turbine, to derive flow characteristics.

Khurana *et al*. [11] used fluent code to analyze the flow in an axial flow hydraulic turbine. They have simulated the flow for both steady and transient state by using a k-ε turbulence model. They concluded that the pressure distribution through the passage is very low on the suction surface but peak velocity is 163 m/s through the passage between consecutive blades.

Materials and methods

The basic concept used in the finite element method (FEM) is that a body or the structure may be divided into smaller elements of finite dimensions called finite elements. The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called nodes or nodal points. The properties of the elements are formulated and combined to obtain the solution for the entire body or structure. The finite element method, sometimes referred to as finite element analysis, according to Hutton is a computational technique used to obtain approximate solutions of boundary value problems in engineering [12]. In early years, applications were performed using mainframe computers which were considered to be very powerful, high speed tools for use in engineering analysis at the time. During the 1960s, the finite element software code NASTRAN was developed in conjunction with the space exploration program in the United States. NASTRAN was the first finite element software code. Among these ANSYS, ALGOR is used in the computational environment. The blades of a Turgo impulse turbine were modeled on Solidworks software (version 2009) [13]. The designs of the blades were converted into IGES format, and they were imported into ANSYS software (Version 14) for meshing [14]. Physical parameters and mechanical properties are shown in **Table 1**. The four tasks involved in the analysis process are:

a. Modeling of the geometry on Solidworks software.
b. Meshing of the geometry on ANSYS.
c. Solving of the equations.
d. Post processing to obtain the results.

FEM Analysis of Turgo Impulse Turbine Blade   Sourabh KHURANA *et al*.

http://wjst.wu.ac.th

Table 1 Physical parameters and mechanical properties of the Turgo impulse turbine.

Serial number	Parameters	Values
1	Nodes	5567
2	Elements	2851
3	Density	8.3e-006 kg mm^{-3}
4	Compressive yield strength	280 (MPa)
5	Tensile yield strength	430 (MPa)
6	Reference temperature	22 °C
7	Poisson's ratio	0.34

In the Turgo impulse turbine, water strikes the blade at an angle of 25 ° [9]. The force transferred by the jet to the blade is presented in this section.

$$\text{Head}(h) = 45 \, m$$

$$\text{Rotational speed}(N) = 1108 \, \text{rpm}$$

$$\text{Flow rate}(Q) = 2.96 \times 10^{-3} \, m^3 / s$$

$$\text{Runner pitch diameter}(D) = 216 \, mm$$

$$V_1 = K_v = 0.98\sqrt{2gh}$$

$$0.98\sqrt{2 \times 9.81 \times 45}$$

$$V_1 = 29.11 \, m/s = Vu_1$$

$$U_1 = \frac{\pi D N}{60} = \frac{\pi \times 216 \times 10^{-3} \times 1108}{60}$$

$$U_1 = 12.53 \, m/s$$

$$Vw_1 = V_1 - U_1$$

$$Vw_1 = 16.58 \, m/s$$

$$Vw_2 = 0.85 \times Vw_1 = 14.093 \, m/s$$

$$Vu_2 = U_2 - Vw_2 \cos 15 = 23.23 \, m/s$$

$$F = \rho \times Q \times (Vu_1 - Vu_2)$$

$$F = 1000 \times 2.96 \times 10^{-3} \times (5.88)$$

$$F = 17.40$$

$$F = 174 \, N$$
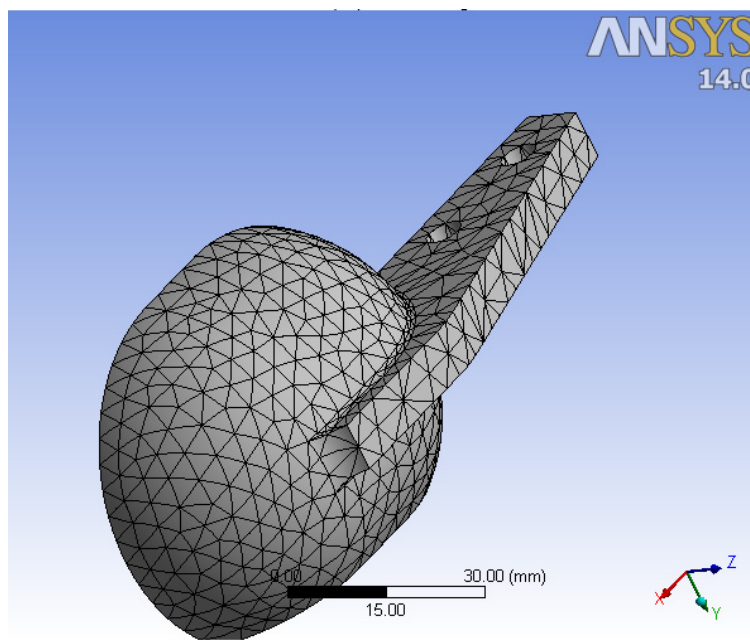
In the calculation, symbols have following descriptions.

Serial number	Symbol	Description
1	V_1	Velocity of Jet at Inlet
2	Kv	Co-efficient of velocity
3	U_1	Tangential velocity of Runner
4	Vw_1	Velocity of Whirl at inlet
5	Vw_2	Velocity of Whirl at outlet
6	Vu_1	Relative velocity at inlet
7	Vu_2	Relative velocity at outlet
8	F	Force on runner

Results and discussion

The blade of the Turgo turbine and meshing of the blade are shown in **Figures 1 - 3**. The maximum VON MISES stresses are found to be 37.5 N/mm^2. The maximum stresses occur at that point where jet strikes the plate. **Figure 4** shows the total deformation when the jet strikes the blade. Total deformation is maximum at that point where water escapes from the blade after doing work. The maximum value of total deformation is 0.001076 mm.

FEM Analysis of Turgo Impulse Turbine Blade　　　　　　　　　　Sourabh KHURANA *et al*.

http://wjst.wu.ac.th
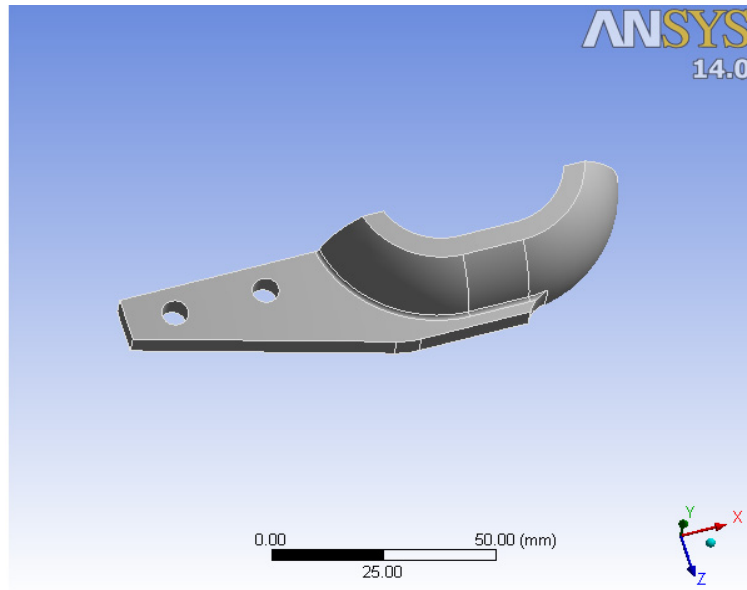
Figure 1 Blade of a Turgo turbine.

Figure 2 Meshing of a Turgo blade.

FEM Analysis of Turgo Impulse Turbine Blade   Sourabh KHURANA *et al*.

http://wjst.wu.ac.th
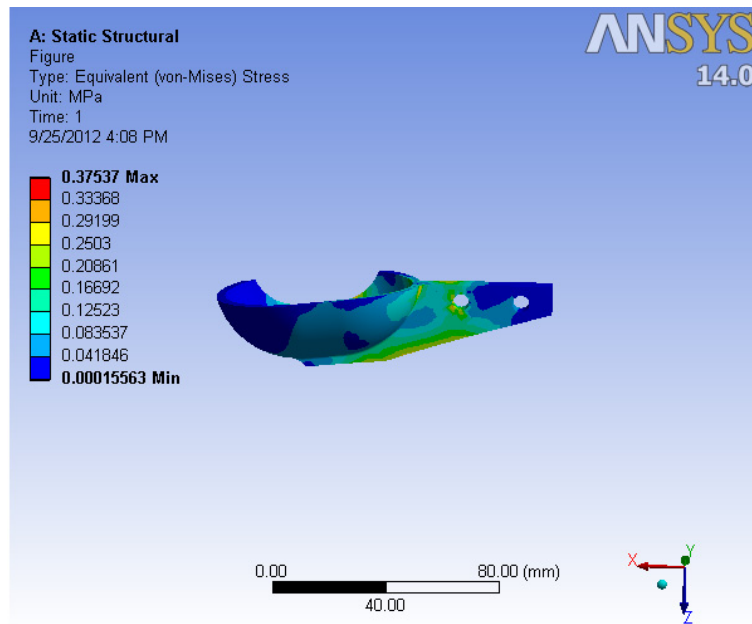
Figure 3 Equivalent stress on the Turgo blade.
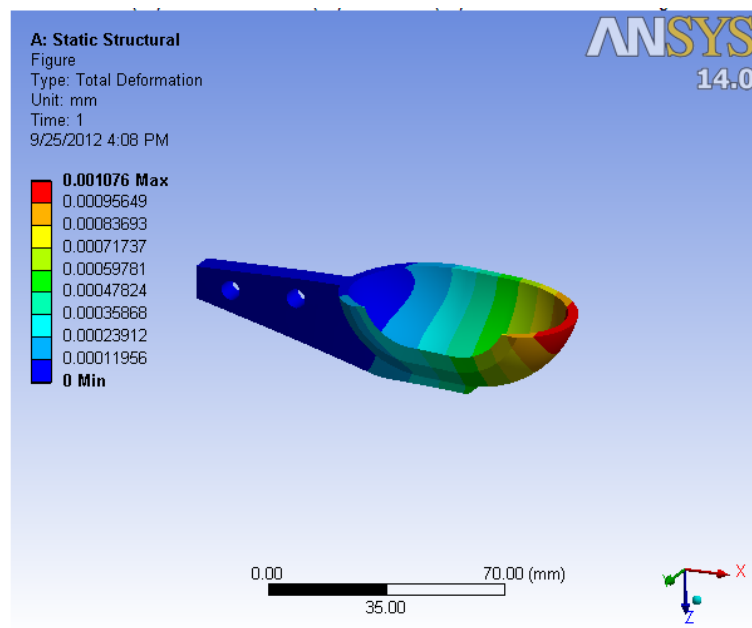
Figure 4 Total deformation on the Turgo blade.

FEM Analysis of Turgo Impulse Turbine Blade   Sourabh KHURANA *et al*.

http://wjst.wu.ac.th

Conclusions

The stress analysis and deformation on the Turgo turbine blade has been carried out by using ANSYS software. Maximum VON MISES stresses are 37.5 N/mm^2, while the deformation produced when the jet strikes the blade is 0.001076 mm. At a force of 174 N, the design made on Solidworks software is safe.

Acknowledgement

References

[1] Ministry of power government of India, Available at: http://www.powermin.nic.in/ JSP_SERVLETS/internal.jsp, accessed October 2012.

[2] BR Cobb and KV Sharp. Impulse (Turgo and Pelton) turbine performance characteristics and their impact on pico-hydro installations. *Renew. Energ.* 2013; **50**, 959-64.

[3] JS Anagnostopoulos and DE Papantonis. Flow modeling and runner design optimization in turgo water turbines. *World Acad. Sci. Eng. Tech.* 2007; **28**, 206-11.

[4] P Alexandre. *Hydrodynamics of the Free Surface Flow in Pelton Turbine Buckets. In*: Lausanne: EPFL. 2007, p. 1-124.

[5] BS Thapa and OG Dahlhaug. Empirical modelling of sediment erosion in Francis turbines. *Energy* 2012; **41**, 386-91.

[6] HP Neopane, OG Dhalhaug and B Thapa. Alternative design of a Francis turbine for sand laden water. *In*: Proceedings of the International Conference on Hydro-power-Hydro, Sri Lanka, 2007.

[7] Y Nakanishi, F Tsuneaki and S Kawaguchi. Numerical and experimental investigations of the flow in a stationary Pelton bucket. *J. Fluid Sci. Tech.* 2009; **4**, 490-9.

[8] J Vesely and M Varner. A case study of upgrading of 62.5 MW Pelton turbine. *In*: Proceedings of the International Conference IAHR 2001, Chez Republic, 2001.

[9] PK Koukouvinis, JS Anagnostopoulos and DE Papantonis. SPH method used for flow predictions at a Turgo impulse turbine: Comparison with fluent. *World Acad. Sci. Eng. Tech.* 2011; **79**, 659- 66.

[10] R Kharei, V Prasad and S Kumar. CFD approach for flow charactersitics of hydraulic Francis turbine. *Int. J. Eng. Sci. Tech.* 2010; **3**, 824-31.

[11] S Khurana, A Kumar and S Kumar. CFD analysis of axial flow hydraulic turbine. *In*: Proceedings of the 4th International Conference on Advances in Mechanical Engineering NIT, Surat, India, 2010.

[12] DV Hutton. *Fundamentals of Finite Element Analysis*. Tata McGraw Hill, Delhi, India, 2005.

[13] Solid Works software (version 2009), Dassault Systems SolidWorks Corporation. USA.

[14] ANSYS Software (Version 13.0), ANSYS Software Private Limited, Inc., USA.