

A Review on Analysis of Low Pressure Stage of Steam Turbine Blade with FEA (ANSYS Software)

Kinnarrajsinh P. Zala¹ Dr. K. M. Srivastava² Nilesh. H. Pancholi³

¹ME Scholar ^{2,3}Professor

^{1, 2, 3} IITE-Indus University, Ahmedabad, India

Abstract— Steam turbine blades are one of the most critical components in power plants. Blade is a major component of the turbine, which receives the impulse directly from the steam jet and converts this force into the driving force. Statistics has shown that LP blades are usually more predisposed to failure compared to blades in HP or IP turbines. The present research work analyses the effects of thermal and structural load on a steam turbine blade under the operating conditions. Stresses due to thermal and dynamic loads of low Pressure Steam Turbine blade of 210 MW power stations analyzed in two stages. In first stage a three dimensional model of turbine blade was prepared in CRE 2.0.

This model will import in ANSYS-14.5 for Finite Element Analysis. Maximum stress and stress distribution is compute using Finite Element Analysis (FEA) at the corresponding section.

Key words: Low pressure Blade, Steam Turbine, Stress Distribution, and Finite Element Analysis by ANSYS software.

I. INTRODUCTION

Steam turbines are major prime movers in thermal power stations. The main parts of simple impulse steam turbine are rotor, blades and nozzles. Turbine blade is exposed to various loads such as thermal, inertia, and bending and may fail due to different factors like Stress-Corrosion Cracking, High-Cycle Fatigue, Corrosion-Fatigue Cracking, Temperature Creep Rupture, Low-Cycle Fatigue, corrosion, etc.

The software offers a comprehensive range of stress analysis and other capabilities in an integrated package for such large-scale, complex problems. An integrated infrastructure, ANSYS Parametric Design Language customization capabilities and nonlinear simulation with contact plasticity work together to provide powerful simulation capabilities for this type of application. Key dimensions of the blade root were modified using ANSYS Parametric Design Language (APDL) capabilities, with ANSYS Mechanical software analyzing the various combinations of parameters.

Ever since the evolution of FEA, there has been a continuous and growing need for a powerful design analysis tool in the power generation industry. In general, turbines represent a class of challenging mechanical prime movers where steady and transient stresses (mechanical and thermal), turbine blade vibrations, and the start/stop cycling of the machines present interesting design challenges to produce a highly reliable machine with long design lives.

These design lives may be as long as 20-25 years (150,000 to 200,000 hours of operation) for steam turbines, and as short as 3 years (25,000 hours) for certain components in a gas turbine. Particular interest is the analysis of turbine blades, as these rotating components, if the separated from their attachment to the rotor, have the potential for causing a tremendous amount of consequential damage, both in the form of human life and property damage.

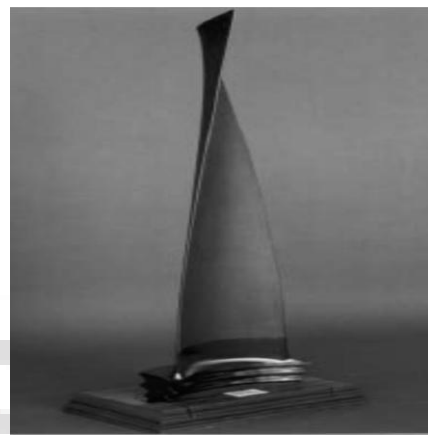


Fig. 1: LP Turbine Blade

A. Problem Definition

All modern steam power plants use impulse-reaction turbines as their blading efficiency is higher than that of impulse turbines. Last stage of steam turbine impulse-reaction blade are very much directly affect efficiency of plant. With the information that an understanding of the forces and stresses acting on the turbine blades is vital importance, in this work we will compute such a force acting on a last stage Low Pressure (LP) blade of a large steam turbine rotating at 3000 rpm in order to estimate the material stresses at the blade root. One such LP steam turbine blade is show in Figure 1. We studied structural and thermal analysis of blade using FEA for this work and by use of the operational data have performed by using FEA (ANSYS) and This study work involved the analyze blade and check FEA data of std. blade with various material.

B. Objective

The main objectives of the study were

- 1) To study effect of stress distribution on steam turbine blade
- 2) structural and thermal analysis of steam turbine blade
- 3) Calculating stresses through advanced computer modelling techniques and simulation.

- 4) To study effect of pressure and get the response of the blade.

II. RESEARCH METHODOLOGY

The following methodology used for carrying out Finite Element Analysis of 210 MW low-pressure blade of Steam turbine.

- 1) Formulation of the problem – the success of any experiment is dependent on a full understanding of the nature of the problem.
- 2) Validation of FEM method for finding maximum stress.
- 3) Modeling of the steam turbine blade will done using CAD software CREO 2.0.
- 4) Stress distribution on turbine blade and root.
- 5) Software ANSYS 14.5 using for Finite Element Modeling.

A. Validation of fem method for finding stress

Steps of validation of fem method for finding Stress:

- 1) Prepare a three dimensional model in CREO 2.0.
- 2) Import the CREO2.0 model in ANSYS software.
- 3) Mesh the ANSYS model.
- 4) Apply boundary conditions and find out stress distribution in the component.
- 5) Validate that maximum stress in component with operating condition

III. LITERATURE REVIEW

Many investigators have suggested various methods to explain the effect of stress and loading on turbine blade, roter and analysis the various parameters:

John. V, T. Ramakrishna was investigated on design and analysis of Gas turbine blade, CATIA is used for design of solid model and ANSYS software for analysis for F.E. model generated, by applying boundary condition, this paper also includes specific post processing and life assessment of blade. How the program makes effective use of the ANSYS pre-processor to mesh complex geometries of turbine blade and apply boundary conditions. The principal aim of this paper is to get the natural frequencies and mode shape of the turbine blade. In this paper we have analyzed previous designs and generals of turbine blade to do further optimization, Finite element results for free standing blades give a complete picture of structural characteristics, which can utilized for the improvement in the design and optimization of the operating conditions.

Subramanyam Pavuluri, Dr. A. Siva Kumar was investigated on design of high pressure steam turbine blade addresses the issue of steam turbine efficiency. A specific focus on airfoil profile for high-pressure turbine blade, and it evaluates the effectiveness of certain Chromium and Nickel in resisting creep and fracture in turbine blades. The efficiency of the steam turbine is a key factor in both the environmental and economic impact of any coal-fired power station. Based on the research presented modifications to high-pressure steam turbine blades can made to increase turbine efficiency of the turbine. The results and conclusions are presented for a concerning the durability problems experienced with steam turbine blades. The maximum

operational Von Mises Stresses are within the yield strength of the material but the deformation is comparatively better for material CA-6 NM (Chromium Nickel). Modified solutions for Steam turbine blade values to machines to maximize their reduce life cycle costs, efficiency, and improve reliability

Sanjay Kumar was investigated on creep life of turbine blade. Inertia load is the constant load that will cause creep failure. Creep is a rate dependent material nonlinearity in which material continues to deform in nonlinear fashion even under constant load. This phenomenon is predominant in components, which exposed to high temperatures. By studying the creep phenomenon and predicting the creep life of the component, we can estimate its design life. The main objective is to predict the creep life of the simple impulse steam turbine blade, and to give the FEM approach for creep analysis. The analysis of turbine blade for different loads, which shows that the maximum stresses, induced in each case. These stresses are within yield limit of the material and will not undergo plastic deformation during operation result is found that, creep life decreases as the stress value increases. Hence, by decreasing the stress value in the component we can increase its creep life. This was achieved by modifying the blade design.

Avinash V. Sarlashkar, MARK L. Redding investigated on the architecture and capabilities of Blade Pro. An ANSYS based turbine blade analysis system with extensive automation for solid model and F.E. model generation, boundary condition application, file handling and job submission tasks for a variety of complex analyses; the program also includes turbo machinery specific post-processing and life assessment modules. Blade Pro is a cutting-edge example for vertical applications built on the core ANSYS engine using ANSYS APDL. Examples of how the program makes effective use of the ANSYS preprocessor to mesh complex geometries of turbine blade and apply boundary conditions are presented using specific examples. A real world application is used to demonstrate the pre-processing capabilities, static and dynamic stress analyses results, generation of Campbell and Interference diagrams and life assessment. The principal advantage of Blade Pro is its ability to generate accurate results in a short amount of time, thus reducing the design cycle time. The good correlation achieved is a testament to the accuracy of the ANSYS solvers and validity of the modeling techniques adopted in Blade Pro.

DR.SHANTHARAJA.M, DR. Kumar. K., was work on the large variety of turbo-machinery blade root geometries used in industry prompted the question if an optimum geometry could be found. An optimum blade root was defined, as a root with practical geometry which, when loaded returns the minimum fillet stress concentration factor. The present paper outlines the design modification for fillet stresses and a special attention made on SCF of the blade root (T-root) which fails and to guarantee for safe and reliable operation under all possible service conditions. Finite Element Analysis is used to determine the fillet stresses and Peterson's Stress Concentration Factor chart is effectively utilized to modify the blade root. The root modified due to the difficulty in manufacturing the butting surface of the tang that grips the blade to the disk crowns

having small contact area. Verify the same using Finite Element Analysis for two cases with and without the tang in the blade. Firstly, to study the fillet stresses with tang and then Peterson's chart is used to reduce the peak stresses with the modification to the butting area and reducing the fillet radius. To conduct the sensitivity analysis for the fillet stresses in blade and disk using FEA.

In the work, the first stage rotor blade of the gas turbine has analyzed using ANSYS 9.0 for the mechanical and radial elongations resulting from the tangential, axial and centrifugal forces. The gas forces namely tangential, axial were determined by constructing velocity triangles at inlet and exist of rotor blades. The material of the blade was specified as N155. This material is an iron based super alloy and structural and thermal properties at gas room and room temperatures the turbine blade along with the groove blade is modeled with the 3D-Solid Brick element. The geometric model of the blade profile is generated with splines and extruded to get a solid model in CATIA. The first stage rotor blade of a two-stage gas turbine has been analyzed for structural, thermal and modal analysis using ANSYS 9.0 Finite Element Analysis software. The gas turbine rotor blade model meshed in HYPERMESH 7.0, meshing software. The thermal boundary condition is such as convection and operating temperatures on the rotor blade obtained by theoretical modeling. Analytical approach used to estimate the tangential, radial and centrifugal forces.

In this work the failure of a second stage blade in a gas turbine was investigated by metallurgical and mechanical examinations of the failed blade. The blade was made of a nickel-base alloy Inconel 738LC. The turbine engine has been in service for about 73,500 hrs. Before blade has failure. Due to the blade failure, the turbine engine was damaged severely. The investigation started with a thorough visual inspection of the turbine and the blades surfaces, followed by the fractography of the fracture surfaces, micro structural investigations, chemical analysis and hardness measurement. The observation showed that a serious pitting occurred on the blade surfaces and there were evidences of fatigue marks in the fracture surface. The micro structural changes were not critical changes due to blade operation at high temperature. It found that the crack initiated by the hot corrosion from the leading edge and propagated by fatigue and finally, because of the reduction in cross-section area, fracture was completed. An analytical calculation parallel to the finite element method utilized to determine the static stresses due to huge centrifugal force. The dynamic characteristics of the turbine blade evaluated by the finite element mode and harmonic analysis. Finally according to the log sheet records and by using a Campbell diagram there was a good agreement between the failure signs and FEM results which showed the broken blade has been resonated by the third vibration mode occasionally before the failure occurred.

IV. CONCLUSION

From above literature, review it is indicated that

- 1) There are many researches done on turbine blade in high-pressure stage or rotor section and work also done in steam and gas based power plant. But I found that are very few researches done work on last stage

tangent-twisted blade of LP stage of turbine so we want to do research on this section. We like to use FEA for analysis.

- 2) Turbine blade tupe,flow of steam through impluse or rection blade,geomeatry of blade,helix angle,force on the blade,material properties of the blade,speed of the turbine and etc. are various parameter important for the turbine blade condition.
- 3) They affect the blade life and efficiency of the plant in terms of mechanical properties and working codition.
- 4) The complex Design of blade will done in CREO 2.0 parametric software with coordinate point of tip and root section of blade
- 5) To perform analysis with appropriate set of parameters is done in ANSYS14.5 software (FEA).

V. SCOPE OF THE WORK

- 1) The turbine operation parameter and effect of steam and force acting on blade can be predict so if want to varies input the parameter can directly predict the effect of output by using FEA.
- 2) The condiiton of the blade in working parameter of turbine and which amount of load will capable by turbine blade can predict by the FEA.

REFERENCES

- [1] John. V, T. Ramakrishna, "The Design And Analysis Of Gas Turbine Blade", International Journal of Advanced Engineering Research and Studies, VOL,2., ISSUE I,OCT.-DEC.,2012.page no,53-55
- [2] Subramanyam PAVULURI, DR. A. Siva KUMAR, "Experimental Investigation On Design Of High Pressure Steam Turbine Blade", International Journal of Innovative Research in Science, Engineering and Technology,VOL. 2, ISSUE 5, MAY 2013, ISSN: 2319-8753
- [3] S. M. Sanjay Kumar, "Creep Life Prediction of Steam Turbine Blade Using Finite Element Method." International Journal of Advanced Engineering Research and Studies, VOL. I, NO.2, PP. 95-98
- [4] Avinash V. Sarlashkar ,MARK L. Redding, " An Ansys-Based Turbine Blade Analysis System" Impact Technologies, LLC, ROCHESTER, NY 14623, U.S.A
- [5] DR.SHANTHARAJA.M, DR. Kumar. K. "Design Modification For Fillet Stresses In Steam Turbine Blade", IJAE International Journal of Advanced Engineering Research and Technology, VOL III,ISSUE I 2012
- [6] G.NARENDRANATH, S.SURESH,"THERMAL Analysis Of A Gas Turbine Rotor Blade By Using Ansys", International Journal of Engineering Research and Applications, VOL. 2, ISSUE 5, SEPTEMBER-OCTOBER 2012, PP.2021-2027, ISSN: 2248-9622
- [7] Patil A.A.,SHIRSAT U.M., "Study Of Failure Analysis Of Gas Turbine Blade", IOSR Journal of Engineering, ISSN: 2250-3021, ISBN: 2878-8719, PP 37-43