

DESIGN & MANUFACTURING OF STEAM TURBINE BLADE

H.KAMALASREE*

Department of S&H. MLR Institute of Technology
Hyderabad-500043, Telangana, India kamalasinivasu@gmail.com

A.V.LAXMAN RAO²

Department of S&H. MLR Institute of Technology
Hyderabad-500043, Telangana, India.

Abstract

A turbine is a device that converts chemical energy into mechanical energy, specifically when a rotor of multiple blades or vanes is driven by the movement of a fluid or gas. In the case of a steam turbine, the pressure and flow of newly condensed steam rapidly turns the rotor. This movement is possible because the water to steam conversion results in a rapidly expanding gas. As the turbine rotor turns, the rotating shaft can work to accomplish numerous applications, often electricity generation.

Steam Turbine blade is designed with a significant factor of safety, the operating forces and stresses calculated above are normal for a last stage blade and under normal conditions this should not pose any risk to the blades integrity. Large steam turbine blades are manufactured from 12% chrome high alloy steels where maximum design stress values of 200-300 N/mm² are permissible. This paper demonstrates that under normal operating conditions, design and manufacture and maintenance strategy are to ensure safe operation.



Fig : Front view of Blade Fig : Back view of Blade

Key Words : vanes, numerous, integrity

INTRODUCTION

Electrical power has become an essential facet of life in modern society. In the home, electricity is used to power water pumps that provide running water, and heating and air conditioning systems that provide a comfortable atmosphere. Electricity is used to power the network of computer systems used extensively in

commercial inventory organization, and in the computerized mechanisms that govern manufacturing processes in the manufacturing industry. These are just a few examples that indicate how much modern society relies on electrical power and how important it is to society that a reliable source of electricity be established.

Currently, nearly 90% of the electricity produced in the United States is generated using steam turbines, while about 80% of that produced worldwide comes from steam generators. With such a substantial amount of electrical being produced by steam turbine generators, it is in the best interest of society to make these generators as efficient and as sustainable as possible. One of the key factors influencing the efficiency of these turbines is the design of the turbine blades. It is through a century of development and advancement in steam turbine blade design that steam turbine efficiency rose from a mere 60% to 90% or better. Thus, the better the design of the blade, the more efficient the turbine will be; and the more efficient the turbine is, the more efficient the generator is and the better off society is. Therefore, it is beneficial to society to implement advanced blade technology that enhances turbine efficiency.

Blades are the heart of a steam turbine, as they are the principal elements that convert the thermal energy into kinetic energy. The efficiency and reliability of a turbine depend on the proper design of the blades. It is therefore necessary for all engineers involved in the steam turbines engineering to have an overview of the importance and the basic design aspects of the steam turbine blades. Blade design is a multi-disciplinary task. It involves the thermodynamic, aerodynamic, mechanical and material science disciplines. A total development of a new blade is therefore possible only when experts of all these fields come together as a team.

Efficiency of the turbine is depends on the following parameters.

- Inlet and outlet angle of the blade.
- Blade Materials.
- Profile of the blade.
- Surface finishing of the blade.

The most critical aspect of steam turbine reliability centers on the bucket design. Since buckets, or rotating blades, are subjected to unsteady steam forces during operation, the phenomenon of vibration resonance must be considered. Resonance occurs when a stimulating frequency coincides with a natural frequency of the system. At resonance conditions, the amplitude of vibration is related primarily to the amount of stimulus and damping present in the system. High bucket reliability requires designs with minimum resonant vibration. The design process starts with accurate calculation of bucket natural frequencies in the tangential, axial, torsional, and complex modes, which are verified by test data. In addition, improved aerodynamic nozzle shapes and generous stage axial clearances are used to reduce bucket stimulus. Bucket covers are used on some or all stages to attenuate induced vibration. These design practices, together with advanced precision manufacturing techniques, ensure the necessary bucket reliability. Almost all of the blading used in modern mechanical drive steam turbines is either of drawn or milled type construction. Drawn blades are machined from extruded airfoil shaped pieces of material stock. Milled blades are machined from a rectangular piece of bar stock.

Experimental evolution

Rotating blades have been recognized as one major cause of failure in many turbine and jet engines. They are, usually rotating at high speeds, interacting with the erosive environment, have complicated shapes, and undergo server dynamic and thermal loadings. These operating conditions expose blades to many vibration excitation mechanisms and the same time make the vibration measurement process of blades a very complicated task. Experiments are done to evaluate the frequencies of blades.

Rao.J.S et.al [13] has done experiment to determine overall damping in a rotating turbine disc-blade system. A test spin rig has been designed and fabricated. Transient excitation of rotating blades is caused by

suddenly shutting off the excitation to the rotating blades. Frequency analysis of the transient blade response gives the information about the modal damping with the speed of rotation and strain amplitude is obtained.

BLADE DESIGN FACTORS

Based on the information provided in the previous sections, an efficient and reliable blade design will satisfy the following requirements:

1. The blade material must have a sufficient yield strength to resist plastic deformation, and must be able to retain such yield strength, or at least most of it, at elevated temperatures.
2. The blade material must be able to be processed and worked easily (This is one of the downsides of Titanium alloys as they are not easily welded and they are expensive to produce).
3. Blade materials must exhibit a moderate elastic modulus so that the blade neither deforms exceedingly nor breaks suddenly under normal operating stresses.
4. Preferably, blade materials should be low-density in order to decrease the centrifugal forces, and therefore centrifugal stresses, on the blades.
5. The blades must be corrosion-resistant, even in the presence of aggressive ionic solutions formed by impurities in the steam.
6. Blades must be manufactured in such a way as to minimize initiation of cracks during the manufacturing process.

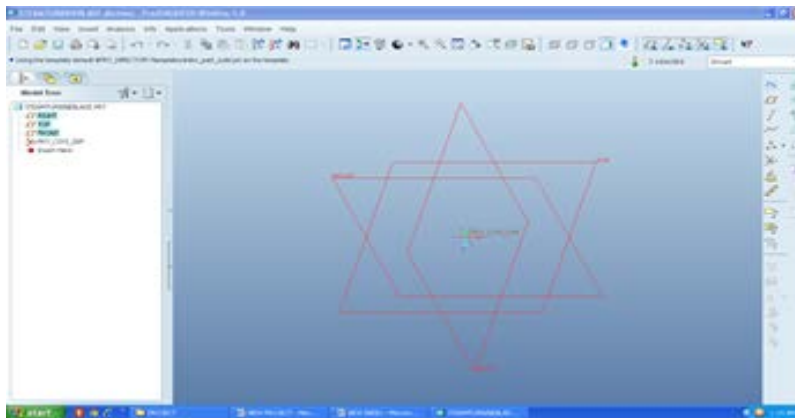


Fig 5.1 The geometry under consideration is generated in the SURF CAM modeling package.

- Click on insert > extrude> placement>sketch/Select plane to sketch> click ok
- Points to be plotted with the required dimensions in X-axis and Y-axis to get the required shape.Check all points and extrude the object to required height .

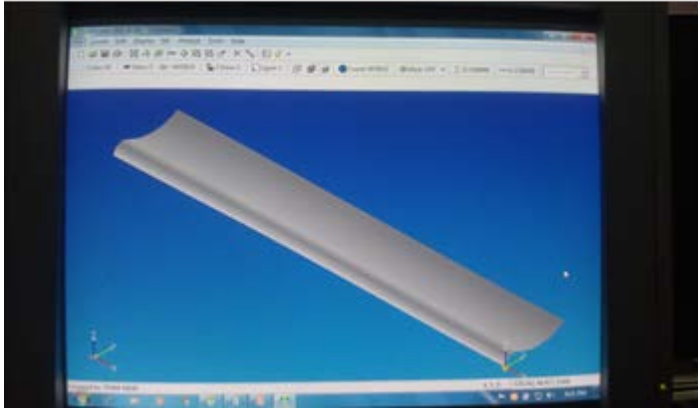


Fig 5.2 Blade Airfoil design by using SURF CAM

- Again select the plane to extrude the I-section on top of the blade
- Extrude the I-section to complete the required shape.

Fig 5.3 Wireframe model of blade

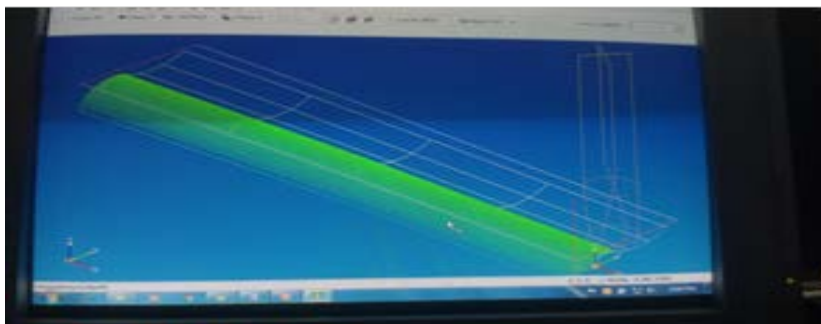
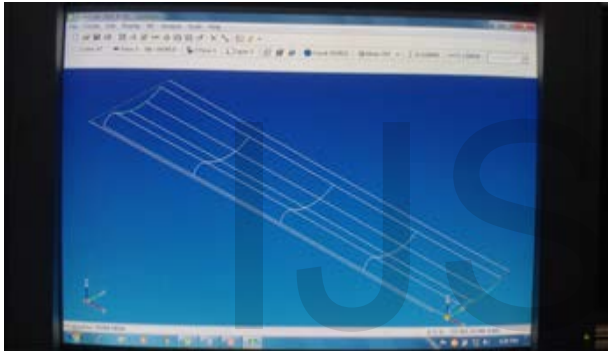


Fig 5.4 Steam turbine blade generated in SURF CAM

- The PRO-E file is saved in *.igs format.
- “*.igs” format is good format for the data translation between any packages.

5.2 INTRODUCTION TO ANSYS

Ansys is general-purpose finite element analysis software, which enables engineers to Perform the following tasks:

1. Build computer models or transfer CAD model of structures, products, components or systems.

2. Apply operating loads or other design performance conditions.
3. Study the physical responses such as stress levels, temperatures distributions or the impact of electromagnetic fields.
4. Optimize a design early in the development process to reduce production costs.
5. A typical ANSYS analysis has three distinct steps
6. Pre processor (build the model).
7. Solution (apply loads and obtain the solution) and Post processor (review the results)

ANSYS Derived Products

In addition to the Ansys /multi physics program and its add on capabilities, a series of ANSYS derived products are available. These products are subsets of ANSYS derived directly from the ANSYS/multi physics program.

ANSYS/mechanical designed for linear and non-linear, structural and thermal, static and dynamic transient analysis. It enables users to solve a wide variety of analysis in mechanical and civil engineering applications. As mentioned previously, ANSYS/mechanical has the linear stress, structural, dynamic analysis, buckling, and heat transfer, thermal, acoustics and Piezo-electric capabilities of ANSYS/mechanical but excludes the three add on options: electro-magnetic, LS-DYNA explicit dynamics, and FLOTRAN computational fluid dynamics (CFD).

ANSYS/Structural product supports the following types of structural analysis, structural static, model, harmonic response, transient dynamic, spectrum, buckling, non-linear structural and p method structural static analysis. Useful for many civil and mechanical engineering applications, the product also enables you to solve fracture mechanics problems and model composites and to perform fatigue evaluations. The ANSYS/structural product does not include the LS-DYNA explicit dynamics, thermal, Electro-magnetic, CFD, acoustics or piezoelectric capabilities of ANSYS.

ANSYS/Linear plus product enables you to do linear static and dynamic structural analysis. Dynamic analysis includes model, harmonic, transient and spectrum analysis. Although it is limited mostly to linear structural solutions. ANSYS/ linear plus has come non-linear capabilities such as large deflection and stress stiffening for some elements and node to node contact elements.

ANSYS/Thermal has steady state and transient analysis capabilities. ANSYS/thermal allows for combined thermal-electric analysis for the elements supported in the product, it provides solution capabilities for a variety of mechanical and electrical engineering Applications.

ANSYS/E-mag is an electromagnetic field simulation product designed for static and low frequency electro-magnetic, electrostatics, conduction, circuit simulation, and coupled circuit electro-magnetic simulation. You can use the 3D version for both three- dimensional and two -dimensional models. The 2D version supports only two-dimensional (planar and ax symmetric) models. When combined with an ANSYS, Structural or thermal product, ANSYS/E-mag also enables you to do coupled magnetic-thermal analysis (such as induction heating) and magnetic structural analysis (such as armature motion). If the add on FLOTRAN CFD capability is available, you can also simulate electro-magnetic CFD coupling.

ANSYS/FLOTRAN is a CFD (Computational Fluid Dynamics) product for fluid flow and heat analysis. It has the same capabilities as add on FLOTRAN CFD capability described above.

ANSYS/Prep post is a pre processing and post-processing product designed for building large models and moving them to different, usually more powerful, computer for solution. It has all the pre processing and post-processing capabilities of ANSYS/Multi physics.

ANSYS/ED is an educational version of ANSYS designed for corporate training programs, academic institutions, and self study. It has all the capabilities of ANSYS/Multi physics and ANSYS/ED does not include composites, and you cannot re link the product to include user features.

Performing a Typical ANSYS Analysis

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, non-linear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analysis for different engineering disciplines. The next few sections of this chapter cover general steps that are common to most analysis.

A typical ANSYS analysis has three distinct steps:

- i. Build the model
- ii. Applying loads and obtain the solution
- iii. Review the results

i. BUILDING A MODEL

Building a finite element model requires more of an ANSYS user's time than any other part of the analysis. First job name and analysis titles are specified. Then the PREP7 pre processor is used to define the element types, element constants, material properties and the model geometry.

Specifying a job name and analysis title:

The task is not required for an analysis, but recommended.

Defining the job name

The job name is the name that identifies the ANSYS job. When you define a job name for an analysis, the job name becomes the first part of the analysis creator, (the extension or suffix for these file names is a file identifier such as "*.db"). By using a job name for each analysis that no files are over written. If the job name is not identified, all files receive the name FILE or file, depending on the operating system.

Defining units

The ANSYS program does not assume a system of units for analysis except in magnetic field analysis; any system of units can be used (units must be consistent for all input data).

Defining Element Types

The ANSYS element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category BEAM3, PLANE77, SOLID96 etc.

The element type determines, among other things:

Degree-of-freedom set (which in turn implies the discipline – structural, thermal, magnetic, quadrilateral, electric, brick etc.). Whether the element lies in two-dimensional or three-dimensional space.

Defining element real constants

Element real constants are properties that depend on the element type, such as cross sectional properties of beam element. For example, real constants are BEAM3, the 2D beam elements are area (AREA), moment of inertia (IZZ), height (HEIGHT), and shear deflections constrain (SHEARZ), initial strain (ISTRIN), and added mass per unit length (ADDMAS). Not all element types required real constants, and different elements of the same type may have differential real constants value.

Defining material properties

Most element types require material properties. Depending on the application material properties may be:

Linear or non linear

Isotropic, orthotropic or an isotropic

Constant temperature or temperature dependent.

As with element type and real constants, each set of material properties has a material reference number. The table of reference numbers versus material property sets is called the material table. Within one analysis, you may have multiple property sets (to correspond with multiple materials used in the model). ANSYS identifies each set with a unique reference number.

Linear material properties: Linear material properties can be constant or temperature dependant and isotropic or orthographic.

Non linear material properties

Non-linear material properties are usually tabular data, such as plasticity data, magnetic field data, creep data, swelling data, hyper elastic material data, etc.

Creating the Model Geometry

Once material properties have defined, the next step in analysis is generating finite element model-nodes and elements that adequately describe the model geometry.

There are two methods to create the finite element method:

Solid modeling

Direct modeling

With solid modeling we describe the geometric shape of model and instruct the ANSYS program to automatically mesh, the geometry with nodes and elements. With direct generation, the location of each node and connectivity of each element can be defined manually.

Applying loads and obtains the solution.

Analysis Type: Analysis type is selected based on the loading conditions and the response ANALYSIS types that are performed in the ANSYS program are given below. Static (or steady state), transient, harmonic, modal, spectrum, buckling. Analysis options allow customizing analysis type. Typical analysis options are the method of solution, stress stiffening on or off, and Newton-Rapson options.

ii. APPLYING LOADS

Loads applied include the boundary conditions (constrains, supports, or boundary field specifications) as well as other externally and internally applied loads. Loads in the ANSYS program are divided into 6 categories.

Degree of freedom constrains

Forces

Surface loads

body loads\

Inertia loads

Coupled Field Loads

Most of these loads can be applied either on solid model (key points, lines and areas) or the finite element model (nodes and elements).

Two important load-related terms in ANSYS are load step and sub step. A load step is simply a configuration of loads for which you obtain a solution. In a structural analysis, for example you may apply wind loads in one load step and gravity in second load step. Load steps are also useful in dividing load

history curve into several segments.

Sub steps are incremental steps taken within a load step. You use them mainly for accuracy and convergence purposes in transient and non linear analysis. Sub steps are also known as time steps, steps taken over a period of time. Note the ANSYS program uses the concept of time in transient analysis and static analysis. In a transient analysis, time represents actual time simply acts as a counter to identify load steps and sub steps.

Specifying load step options

Load step options that you can change from load step to load step, such as number of sub steps, time at the end of a load step, and output controls. Depending on the type of analysis you are doing, load step options may be required. The analysis procedures in the analysis guide manuals describe the approximate load step options as necessary.

iii. Solution: In the phase of analysis, the computer takes over and solves the simultaneous equations that the finite element method generates. The results of the solution are:

Nodal degree-of-freedom values, which form the primary solution,

Derived values, which form the element solutions.

The element solution is usually calculated at the elements integration points. The ANSYS program writes the results to the data base as well as to the result file. Several methods of solving the simultaneous equations are available in the ANSYS program: frontal solution, sparse direct solution, Jacobin conjugate gradient (PCG) solution, and an automatic iterative solver option (ITER). The frontal solver is the default, but you can select a different solver.

Post processing

Post processing means reviewing the results of an analysis. It is probably the most important step in the analysis, because you are trying to understand how the applied loads affect your design, how good your finite element mesh is, and so on.

Two post-processors are available to review your results: POST1, the general postprocessor, and POST26, the time history postprocessor. POST1 allow you to review the results over the entire model at specific load steps and sub steps (or at specific time points or frequencies). In a static structural analysis, For example, you can display the stress distribution for load step 3 or in a transient thermal analysis, you can display the temperature distribution at time=100 seconds.

POST26, the time history postprocessor enables you to review analysis results at specific points in the model as a function of time, frequency, etc. it has many capabilities ranging from simple graphics display and tabular listing to more complex operations such as differential calculus and response spectrum generation. A typical usage of POST26 is to graph result items versus time in a transient analysis, or to graph force versus deflection in a non-linear structural analysis. In a transient magnetic analysis, for instance, you can graph the force at a particular node versus its deflections.

It is important to remember that the postprocessor in ANSYS are just tools for reviewing analysis results. You still need to use your engineering judgment to interpret the results. For example, a contour display may show that the highest stress in the model is 37,800 psi. it is now up to you to determine whether this level of stress is acceptable for your design.

The Results File

The ANSYS solver writes results of an analysis to the results file during solution. The name of the result

file depends on the analysis discipline:

- Job name.RST for a structural analysis
- Job name.RSH for a thermal analysis
- Job name.RMG for a magnetic field analysis
- Job name.RFL for a FLOTRAN analysis

For a FLOTRAN analysis, the file extension is .RFL, for other fluid analysis, the file extension is .RST or .RTH, depending on whether structural degrees of freedom are present.(using different file identifiers for different disciplines helps you in coupled-field analysis where the results from one analysis are used as loads for another. This presents a complete description of coupled-field analysis).

Types of Data Available For Post Processing

The solution phase calculates two types of result data:

- ☐ Primary data consist of the degree of freedom solution calculated at each node: displacements I a structural analysis, temperature in a thermal analysis, magnetic potentials in a magnetic analysis, and so on.
- ☐ Derived data are those results calculated from the primary data, such as stresses and strains in a structural analysis, thermal gradients and fluxes in s thermal analysis, magnetic fluxes in magnetic analysis, and the like.

They are typically calculated for each element, at all integration points of each element, or at the centroids of each element. Derived data are also known as element solution data, except when they are averaged at the nodes. In such cases, they become nodal solution data.

Analysis of the model is done in ANSYS software

- i. The model is imported into ANSYS as File>import>.ges.....
- ii. Once file is imported, need to check all volumes and areas for any data loss generally occurs.
- iii. Once model is free from the geometric errors, than save the file as File>save as>filename.db
- iv. *.db is the extension of the ANSYS file
- v. Preferences >structural>ok
- vi. Preprocessor>element type >brick solid45>ok
- vii. Preprocessor>material prop>structural>linear>elastic>isotropic>modulus of Elasticity (2.15e5)>Poisson Ratio(0.3)>ok
- viii. Preprocessor>material prop>structural>density(7.7e-9)>ok
- ix. Meshing>mesh tool > Assign element size(2)>volume mesh
- x. Solution>define loads>apply>Structural >Displacement >on areas> (select appropriate area) >ok
- xi. Solution > define loads > Angular velocity >ok
- xii. Solution >Analysis type >static >per stressed condition on > ok
- xiii. Solution > analysis type > modal Analysis > no of Modes to extract >per-stressed condition on > ok
- xiv. Solution > solve>current L.S >ok
- xv. General post processor >nodal solution > deform shape > def+ unreformed shape
- xvi. General post processor > read results >by pick >gives the set of natural frequencies
- xvii. Once frequencies are known to us, we can read particular frequency and we can go for mode shapes

Frequencies vs. speeds gives the Campbell diagram which is helpful to find out the critical frequencies, thereby we avoid the resonance effect.

CONCLUSIONS

The implementation of robust turbine blades, designed in accordance with the latest material technologies and able to withstand the most trying of circumstances, in combination with the use of clean, renewable fuels presents an efficient method of generating substantial amounts of electricity. An improved blade design, focused on resisting the effects of stresses, corrosive agents, and creep-inducing temperatures, will elevate the turbine efficiency, consequently leading to an increase in the power plant's overall efficiency, a reduction of the amount of fuel consumed, and ultimately a decrease in operating costs. To improve efficient blade design, will serve to reduce operating costs even further and lessen the environmental impacts of steam turbines. Overall, such a combination of technologies would benefit society by providing an efficient, viable, and sustainable means of generating electrical energy.

The results and conclusions are presented for a study concerning the durability problems experienced with steam turbine blades. The maximum operational VonMises Stresses are within the yield strength of the material but the deformation is comparatively better for material AISI 422 (martensitic stainless steel). Modernization solutions are the application of the Steam turbine blade technology to existing machines to maximize their efficiency, improve reliability, and reduce life cycle costs. The discussion of these manufacturing solutions focused on the technology and this project is made on the basic manufacturing technique of steam turbine blades. The procedure involved in this manufacturing leads to achieve the best surface quality and structure. This method uses two separate cutting steps to reach the full depth of cut, which allows the cutting force to be reduced more effectively than by reducing the feed per tooth, as it allows the chip thickness to be modified towards the recommended target values.

Stress analysis is carried out for the root with blade for different speeds here pre stress effects is also included and von mises stress is obtained for 3000 rpm as shown in fig 5.1. And the table 5.2 Natural frequencies for root with blade are obtained from modal analysis. Campbell Diagram is drawn for root and blade by taking the frequencies obtained for different speeds to see whether the blade is resonating in the operating region in table 5.1 and fig 5.2. It is observed that the first two modes are not intersecting the engine order lines as shown in the fig 5.2. Thus they are free from resonance. Only first two bending modes are considered because they are only critical and cause failures in turbine blade.

Blade	RPM	1st MODE	2nd MODE	3rd MODE	4th MODE
250NEWLP3	0	88.668	177.97	361.92	388.22
	1000	93.466	182.61	363.27	392.57
	2000	106.25z	195.64	367.09	405.36
	3000	123.99	214.88	373.1	425.53

3600 136.03 228.37 377.68 440.53

Table 6.1 Natural Frequencies of Different Mode Shapes at Different RPM

S. NO

	Deflection (mm)	VON Mises Stress (MPa)			
		UX	UZ	Minimum	Maximum
1	0	3.90133	906.435	3.90133	906.435
2	0.1	4.20005	907.688	4.15046	922.043
3	1	4.20815	911.019	4.16555	939.074
4	5	3.2306	925.832	3.47329	1014.82
5	10	1.8457	1109.57	2.46175	1437.48

Table 6.2 Von Mises Stresses at 3000 rpm for different Deflection

REFERENCES

- [1] 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 (IJIRSET) ISSN: 2319-8753, Vol. 2, Issue 5, May 2013, pp:1469-1476.
- [2] Rao, J.S., 1974 "Application of variational principle to shrouded turbine blades", Proceedings of 19th cong, ISTAM. pp. 93-97.
- [3] Leissa, A.W., Macbain, J.C., and Keilb, R.E., 1984, "Vibration of twisted cantilever plates summary of provisions. Current studies", Journal of Sound and Vibration, vol. 96(20), pp. 159-167.
- [4] Tsuneo Tsuiji, Teisuke Sueoka, 1990, "Vibrational analysis of twisted thin cylindrical panels by using Raleigh-Ritz method", JSME International Journal, Series iii, vol. 33, pp. 501-505.
- [5] Le-Chung Shiau. Teng – Yuan Wu., October 1997, "Free Vibration of buckled laminated plates by finite element method", Transactions of the ASME, Journal of Vibrations and Acoustics, vol. 111, pp. 635-644.
- [6] Hu, XX., and Tsuiji, T., Jan 7, 1999, "Free Vibrational analysis of curved and twisted cylindrical thin panels", Journal of sound and vibration, vol 219 (1). pp. 63-68.
- [7] Yoo, H.H., Kwak, J.Y., and Chung, J., Mar 2001, "Vibrational analysis of rotating pre twisted blades with a concentrated mass", Journal of sound and vibration, vol 240(5), pp.891-908.
- [8] Park, Jung –Yong., Jung, Yong-Keun., Park, Jong-Jin., Kang, Young-Ho., 2001, "Dynamic analysis method for prevention of failure in the 1st stage low pressure turbine blade with 2 fingers root". Proceeding of SPIE – the international Society for Optical Engineering, vol 4537, pp. 209-212.
- [9] Shah, A.H., Ramsekhar, G.S., and Desai, Y.M., 2002, "Natural vibrations of laminated composite beams

by using fixed finite element modeling”,

Journal of sound and vibration, vol. 257, pp. 635-651.

[10] Rao, J.S., 1993, “Life estimation of turbine blades”, B.H.E.L (R&D), vol. 14-16, pp. 1-11.

[11] Chen, L.W., & Pengwk, 1995, “Dynamic stability of rotary blades with geometric non-linearity, Journal of sound & Vibration, vol. 187, pp. 421-433.

[12] ANSYS 13.0 Theory Reference, ANSYS Corporation, 2011.

[13] T. Tomioka, Y. Kobayashi and G. Yamada “ Analysis of free vibration of rotating “William J. Palm
“ Mechanical Vibration” Wiley ISBN 0-471-
34555-5, 2004.

[14] BHEL R&D, “Correlation of the Theoretical, Experimental Campbell Diagram with Ansys Campbell
Diagram.

[15] N. S. Vyas and J. S. Rao “Fatigue Life Estimation Procedure for a Turbine Blade Under Transient
Loads” Journal of Engineering for Gas

Turbines and Power , Volume 116 , Issue 1 , February 04, 1992; Online April 24, 2008.

[16] Murari P Singh and Terry Mathews “fatigue damage of steam turbine blade caused. by frequency shift
due to solid buildup” proceedings of

Twenty third turbomachinery symposium.pp-107-114.

IJSER