Life Estimation of a Steam Turbine Blade Using Low Cycle Fatigue Analysis

Ms. Aruna Devi Palakurthi  
PG Student,  
Department of Mechanical Engineering,  
Malla Reddy College of Engineering,  
Secunderabad.  

Mr. Dr. D.K. Nageshrwarao  
Professor,  
Department of Mechanical Engineering,  
Malla Reddy College of Engineering,  
Secunderabad.  

ABSTRACT:  
The present work illustrates, 3D finite element analysis (FEA) of low-pressure (LP) steam turbine bladed disk assembly are carried out at a constant speed loading condition. The prime objective is to study structural integrity of bladed disk root with aid of design considerations at design stage. Secondly, design rules are developed for structural integrity of blades and disk considering a factor of safety for material, manufacturing and temperature uncertainties. These design rules are in turn used as design checks with aid of finite element analysis results. Investigations are performed based on Neuber formulae for solving a highly non-linear problem employing linear analysis tool ANSYS 15.0. Local peak stresses at blade and disk root fillet of linear analysis is used to identify the equivalent non-linear stress value by strain energy distribution method for estimating the minimum number of cycles required for crack initiation for low cycle fatigue (LCF) calculations. Design methodology is developed to address the structural integrity of blades at design point and for off-design conditions.

Keywords:  
Turbine blade, Failure analysis, Creo Elements, FEM, Ansys.

INTRODUCTION:  
In this study, first strain-controlled deformation and fatigue life are calculated for Stem turbine blade and then they are compared with ANSYS results. Various approaches to estimating mean stress effects on strain-life analysis are Morrow method and Smith, Watson, and Topper (SWT) method employed here to estimate the fatigue life of steam turbine blade. Morrow using the true fracture strength is a considerable improvement. However, the Morrow expression employing the fatigue strength coefficient $\zeta_f$ may be grossly non-conservative for metals other than steels. The Smith, Watson, and Topper (SWT) method is a reasonable choice that avoids the difficulties. A steam turbine is a device that extracts thermal energy from pressurized steam and uses it to do mechanical work on a rotating output shaft. In this case, the pressure and flow of steam rapidly turns the rotor. The nozzles and diaphragms in a turbine are designed to direct the steam flow into well-formed, high-speed jets as the steam expands from inlet to exhaust pressure. These jets strike moving rows of blades mounted on the rotor. The blades convert the kinetic energy of the steam into rotation energy of the shaft.
CALCULATION:

**Blade design:**
Blade design is very difficult and confidential to every turbine designer, so here only outlining the design by just some important dimensions only.

**Centrifugal force:**
Centrifugal force is directed outwards, away from the centre of curvature of the path. A simplified 2D figure of the blades under discussion is shown in Figure 5.2.

The general equation for centrifugal force is

\[ F = m r \omega^2 \]  

Where \( m \) is the mass of the moving object, \( r \) is the distance of the object from the centre of rotation (the radius of curvature) and \( \omega \) is the angular velocity of the object. In the case under consideration, we need to account for the fact that the mass of the blade is distributed over its length and the radius of curvature also changes along the length of the blade.

Consider a small segment of mass \( \delta m \), of length having width \( \delta r \) at a distance \( r \) from the centre. Then the equation for the centripetal force \( \delta F \) on this small segment is given by:

\[ \delta F = \delta m r \omega^2 \]  

The blades have a cross sectional area \( A \) (mm²) and material density \( \rho \) (kg/mm³). Then we can write the mass of the element as \( \delta m = \rho A \delta r \). Equation (2) can be written as

\[ \delta F = \rho A \omega^2 r \delta r \]

Formally, it can be written as \( dF = \rho A \omega^2 r dr \). Let be the radius of the rotor disc and be the distance between the centre of the rotor disc and tip of the blade. Then, integrating equation (3) along the total length of the blade, the total centrifugal force acting on the blade is given by

\[ F = \rho A \omega^2 \int r dr \]

Numerically:

Calculation of Centrifugal Force: The following data is considered for design and centrifugal force estimation.

Blade speed

\( N = 8000 \) rpm

Blade cross-sectional area

\( A = 165.161 \) mm²

Material density \( \rho = 7850 \times 10^{-6} \) kg/mm³

Blade tip radius \( r_2 = 267.5 \) mm

Blade root radius \( r_1 = 220.5 \) mm

Blade length \( r_2 - r_1 = 47 \) mm

So we can calculate the angular velocity in radians per second as \( \omega = 8000 \times 2 \pi / 60, \omega = 837.75 \) rad/sec.

Substituting all above values in equation (4)

\[ F = 7850 \times 10^{-6} \times 165.161 \times 837.752 \times (267.52 - 220.52) / 2 \times 1000 \]

\[ F = 10,436.2 \text{N} \]

Hence the magnitude if the centrifugal force acting on the blade due to high angular velocity is 10,436N.

Fatigue life calculation: There are many methods to calculate the fatigue life. Based on the available data, accuracy and ease Smith, Watson and Topper (SWT) Mean Stress Correction for Strain Life method used
for the present work. SWT equation for Fatigue Analysis is given below

Where

\[ \sigma_{\text{max}} = \text{Maximum stress} \]
\[ \Delta \varepsilon / 2 = \text{Total strain amplitude} \]
\[ \sigma' \text{ failure} = \text{Fatigue Strength Coefficient or Effective strength} \]
\[ \varepsilon' \text{ failure} = \text{Fatigue ductility coefficient} \]
\[ E = \text{Modulus of Elasticity} \]
\[ 2N_{\text{failure}} = \text{Number of reversals} \]
\[ b = \text{Fatigue strength exponent} \]
\[ c = \text{Fatigue ductility exponent} \]

Intensity of this work is to estimate the fatigue life of the turbine blade. But here number of cycles “N” in the empirical formula is with different powers, which is difficult to calculate directly. So let us go for trial and error method by assuming some values for “Nf”. \( \sigma_{\text{mean}} = (\sigma_1 + \sigma_2) / 2 \)
\( \sigma_a = (\sigma_1 - \sigma_2) / 2 \)
\( \sigma_{\text{max}} = \sigma_{\text{mean}} + \sigma_a \)

<table>
<thead>
<tr>
<th>S.No</th>
<th>( \sigma_1 )</th>
<th>( \sigma_2 )</th>
<th>( \sigma_{\text{mean}} )</th>
<th>( \sigma_a )</th>
<th>( \sigma_{\text{max}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
<tr>
<td>2</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
<tr>
<td>3</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
<tr>
<td>4</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
<tr>
<td>5</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
<tr>
<td>6</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
<tr>
<td>7</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
<tr>
<td>8</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
<tr>
<td>9</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
<tr>
<td>10</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
<tr>
<td>11</td>
<td>540</td>
<td>0</td>
<td>270</td>
<td>270</td>
<td>540</td>
</tr>
</tbody>
</table>

Pre-processor for blade analysis

Step 1: Geometry creation
First step of Ansys is creating the geometry. This can be directly done in design module or else we can import the geometry from other location also. But it should in the format which can be read by ANSYS. Some of that type of formats is IGES, STEP. Here for this analysis geometry file is imported in “STEP” which is exported from the Creo 1.0 (it is modeling software also known as PROE).
Step 2:
Material assignment is the second step for analysis. Because based on the type of material same geometry will give different results even for same working conditions. So after importing the model, it should be assigned the material properties.

Pre-processor for blade analysis
Step 1: Mesh Generation or Ansys model generation. Mesh generation means discretising the model into small element. Finite element itself explains that dividing a complex element into small well known shape. Once model is assigned with material then meshing of the geometry will be done. Then onwards geometry can be named as ansys model. Here tetrahedron mesh is used to generate the meshed model with element size 1mm.

Step 2: Constraining the element
The main aim of any analysis is to get some results by applying some forces on it. So geometry should not be allowed to free moment due to the force application. Hence it need some constrains. Constrain means arresting the motion at some location. That may be fixed, displaced constrain. Here blade is constrained by fixed supports at side of the blade tang. This is due to locking of blade in rotor disk.

Step 3: Application of loads
Centrifugal force is the major force acting on blades. When compared magnitudes of all other forces acting on blade with centrifugal force magnitude they can be negligible. So in this analysis centrifugal force only considered as load of application.

---

**Fig-4 Imported Blade model**

**Fig-5 Material properties of steel**

**Fig-6 Meshed structure of Blade**

**Fig-7 Constrains applied to Blade**

**Fig-8 Load application to steam turbine blade**
6.4 Post processor for blade analysis

**Step 1:** Solving the model, it means allowing the system to run all given conditions like application of constraints and loads.

**Step 2:** Generating the results after proper solving Analyst can able to get the required results. Here Stress developed within the blade and fatigue life estimation is major criteria of project. So get the results by selecting the required options like equivalent stress and Fatigue tool.

Equivalent von mises stresses observed (563 MPa) on the fillet region of the blade as depicted in the figure.

**MODIFICATIONS SUGGESTED:**
Existing blade design is good enough for fatigue life in theoretical calculation. But there is a problem in finite element analysis. In theoretical calculation blade model is getting infinite life (2.438e6). By running ANSYS software existing blade design is getting only 86436 is number of cycles as fatigue life. Modified design of steam turbine blade Here failure of blade mostly occurs in T root. So it requires some modifications to get the infinite life (1e6). By doing some trial and error methods in changing the dimensions of T root. Final a modification is suggested to turbine designer as below.

1. Neck width of the blade is increased by 1mm. i.e. Neck width is modified to 11mm from 10mm
2. Fillet radius of the root is modified to 0.8 mm from 0.5mm
3. Chamfer dimensions of the tang (bottom part of the root) is changed to 1x45oand 2.77x45o from 1.25xa45oand 3x45o respectively.

**FE Analysis of Modified blade**

**Step 1:** Geometry creation
First step of Ansys is creating the geometry. This can be directly done in design module or else we can import the geometry from other location also. But it should in the format which can be read by ANSYS. Some of that type of formats is IGES, STEP. Here for this analysis geometry file is imported in “STEP” which is exported from the Creo 1.0 (it is modeling software also known as PROE).
Step 2:
Material assignment is the second step for analysis. Because based on the type of material same geometry will give different results even for same working conditions. So after importing the model, it should be assigned the material properties.

Fig 13- Material Properties

Processor for modified blade analysis
Step 1: Mesh Generation or Ansys model generation
Mesh generation means discretising the model into small element. Finite element itself explains that dividing a complex element into small well known shape. Once model is assigned with material then meshing of the geometry will be done. Then onwards geometry can be named as ansys model. Here tetrahedron mesh is used to generate the meshed model with element is

Step 2: Constraining the element
The main aim of any analysis is to get some results by applying some forces on it. So geometry should not be allowed to free moment due to the force application. Hence it need some constrains. Constrain means arresting the motion at some location. That may be fixed, displaced constrain. Here blade is constrained by fixed supports at side of the blade tang. This is due to locking of blade in rotor disk.

Step 3: Application of loads
As discussed in literature survey under chapter 2 centripetal forces is the major force acting on blades. When compared magnitudes of all other forces acting on blade with centripetal force magnitude they can be negligible. So in this analysis centripetal force only considered as load of application.

Fig 16 Load application to modified steam turbine blade
Post processor for modified blade

Step 1: Solving the model, it means allowing the system to run all given conditions like application of constrains and loads.

Step 2: Generating the results

After proper solving Analyst can able to get the required results. Here Stress developed with in the blade and fatigue life estimation is major criteria of project. So get the results by selecting the required options like equivalent stress and Fatigue tool.

Fig.17 Equivalent stress of modified steam turbine blade

With the implementation of design modification, the Equivalent von mises stresses observed (345 MPa) on the fillet region of the blade as depicted in the fig above. The stress levels reduced from 563MPa to 345 MPa which will helps in improving the fatigue life.

Fig.18- Fatigue life of modified steam turbine blade

Minimum Fatigue life observed 1e6 and the entire blade is meeting the requirement as depicted in the fig.

RESULTS:

From the ANSYS results of existing blade and modified blade design following comparisons are made for better understanding of improvement in the blade life.

Table 7.4 Comparison of results

<table>
<thead>
<tr>
<th>S.No.</th>
<th>Parameter</th>
<th>Existing Blade</th>
<th>Modified blade</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Equivalent stress (Mpa)</td>
<td>560</td>
<td>345.1</td>
</tr>
<tr>
<td>2</td>
<td>Fatigue life “N”</td>
<td>86438</td>
<td>1000000</td>
</tr>
</tbody>
</table>

CONCLUSION:

This project has attempted to investigate the fatigue response of the steam turbine blade in terms of high cycle fatigue. The goal of the research was to establish the technique of the high cycle fatigue assessment of the HP turbine blade equipped with the T root and to determine the number of startups to the crack initiation of the particular LP blade. Existing blade design is good enough for fatigue life in theoretical calculation. But there is a problem in finite element analysis. In theoretical calculation blade model is getting infinite life (2.438e6). But during run of ANSYS software existing blade design is getting only 86436 is number of cycles as fatigue life. Here failure of blade occurs in T root of blade. So it requires some modifications to get the infinite life (1e6). By doing some trial and error methods in changing the dimensions for T root of the blade Final a modification is suggested to turbine designer which is able to achieve the life of 1e6 cycles as fatigue life.

REFERENCES:

International Journal of Innovative Research in Science, Engineering and Technology Suhas B1, A R Anwar Khan2S


