DESIGN AND ANALYSIS OF GAS TURBINE BLADE USING FINITE ELEMENT METHODS

A Project report submitted in partial fulfilment of requirements for the award of degree of

BACHELOR OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

BY

Lalu Yadav (14011P0307)

Sri Ramoji Sreeja(14011P0316)

Syed Abdul Khader (14011U0301)

Under the esteemed guidance of

Dr M Sreenivasa Rao

DEPARTMENT OF MECHANICAL ENGINEERING,

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY,

KUKATPALLY,

HYDERABAD-500085.

DEPARTMENT OF MECHANICAL ENGINEERING J.N.T.U.H
COLLEGE OF ENGINEERING HYDERABAD-85.

DECLARATION BY THE CANDIDATE

We hereby declare that the project report entitled "DESIGN AND ANALYSIS OF GAS TURBINE BLADE USING FINITE ELEMENT METHODS" is carried out by us during the year 2017-2018 in partial fulfilment of the requirements for the award of BACHELOR OF TECHNOLOGY in MECHANICAL ENGINEERING at the JAWAHARLAL NEHRU TECHNOLOGICAL COLLEGE OF ENGINEERING, HYDERABAD. We have not submitted the same to any other University / Institute for the award of any degree or diploma.

Lalu Yadav (14011P0307)
Sri Ramoji Sreeja(14011P0316)
Syed Abdul Khader (14011U0301)

DEPARTMENT OF MECHANICAL ENGINEERING

J.N.T.U.H COLLEGE OF ENGINEERING HYDERABAD-85.

CERTIFICATE BY THE HEAD OF THE DEPARTMENT

This is to certify that the project report entitled "DESIGN AND ANALYSIS OF GAS TURBINE BLADE USING FINITE ELEMENT METHODS" submitted by Lalu Yadav(14011P0307), Sri Ramoji Sreeja(14011P0316) and Syed Abdul Khader(14011U0301) in partial fulfillment of the requirements for the award of BACHELOR OF TECHNOLOGY in MECHANICAL ENGINEERING at the JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY COLLEGE OF ENGINEERING HYDERABAD, is a work carried out by them under supervision and guidance of Dr M Sreenivasa Rao. To the best of my knowledge, this work has not been submitted to any other University / Institute for the award of any degree or diploma.

Dr J. SURESH KUMAR
Professor, HOD,
Dept. of Mechanical Engineering,
JNTUH College of Engineering,
HYDERABAD

ACKNOWLEDGEMENT

We write this acknowledgement with the great honour, pride and pleasure to pay our respect to all people who enabled us either directly or indirectly in reaching this stage.

We wish to express our sincere thanks to our internal guide Dr M SRINIVASA RAO, Professor, J.N.T.U Hyderabad for his continuous suggestions, guidance and enormous inspiration in carrying out the project.

We sincerely thank, Dr J. SURESH KUMAR, Professor and Head, Dept. of Mechanical Engineering, for his immense support and motivation.

We also thank BHEL(R&D), Hyderabad for providing us with their design data book for important data towards the project.

We take this opportunity to express our sincere thanks to all the Teaching Faculty of Dept. of Mechanical Engineering J.N.T.U College of Engineering, Hyderabad without whose contribution and continuous teaching this would have not been possible. In addition, we also thank Dr PANITAPU BHRAMARA for her out-of-the-way support, vital teaching and contribution towards knowledge of Gas Turbines and it's working.

We are immensely thankful to each and every faculty member of Mechanical Engineering department for their relentless contribution towards successful completion of our course.

We express our sincere thanks to our senior SAI KIRAN for his essential inputs and for clearing all our doubts during the analysis process. At last but also very important we thank all our classmates, Seniors and our Juniors who contributed much with their support, teachings, morals, understandings and many more throughout the course and for the project.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF GRAPHS

DESIGN AND ANALYSIS OF GAS TURBINE BLADE USING FINITE ELEMENT METHODS

ABSTRACT

Turbine is a device designed to convert the heat energy of the fuel into useful work. In the present work the first stage rotor blade gas turbine has been analyzed for structural, thermal, modal and harmonic response using ANSYS software 18.1, which is a powerful Finite Element analyses tool. In the process to get mechanical and thermal stresses, the temperature distribution, natural frequency, amplitude, phase angles in the blade has been evaluated using this software.

The temperature has a significant effect on the overall stress on the rotor blades; it has been felt that a detail study can be carried out on temperature effects to have a clear understanding of the combined mechanical and thermal stresses for these materials.

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 rotor blade was then analyzed using ANSYS 18.1 for the temperature distribution. The convective heat transfer coefficients on the blade surface exposed to the gas have to feed to the software. The convective heat transfer coefficients were calculated using the heat transfer empirical relations taken from the heat transfer design data book. The radial elongations in the blade were also evaluated.

The material of the blade was specified for three materials as Titanium Alloy, Inconel 718 and Structural Steel. This material's structural and thermal properties at gas room and room temperatures were taken from the design data book available in the library of BHEL(R&D), Hyderabad. The turbine blade along with the groove is considered for the static, thermal modal and harmonic response analysis. The geometric of the blade profile is generated with the splines and extrudes to get the solid model.

The thermal boundary conditions such as convection and operating temperatures on the rotor blade are applied on theoretical modeling. Analytical approach is used to estimate the tangential, radial and centrifugal forces. The purpose of this thesis is to study the effect of stresses, temperature and natural frequencies on gas turbine blades and also compare the same for three different materials to come up with best suitable material.

I. INTRODUCTION

1.1. INTRODUCIION TO THE FINITE ELEMENT METHOD

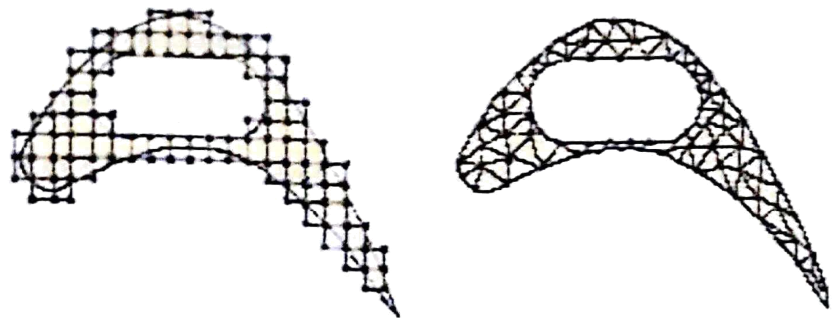
The finite element method is a numerical method that can be used for the accurate solution of complex engineering problem. It is considered to be one of the best methods for solving a wide variety of practical problems efficiently. It is method (FEM) has now become a very important tool of engineering analysis. Its versatility is reflected in its popularity among engineers and designers belonging to nearly all the engineering disciplines. Whether a civil engineer designing bridges, dams or a mechanical engineers designing auto engines, rolling mills, machine tools or an aerospace engineer interested in the analysis of dynamics of an aero plane or temperature rise in the heat shield of a space shuttle or a metallurgist concerned about the influence of a rolling operation on the microstructure of a rolled product or an electrical engineer interested in analysis of the electromagnetic field in electrical machinery-all find the finite element method handy and useful. It is not that these problems remained unproved before the finite element method came into vogue; rather this method has become popular due to its relative simplicity of approach and accuracy of results. Traditional methods of engineering analysis, while attempting to solve an engineering problem mathematically, always try for simplified formulation in order to overcome the various complexities involved in exact mathematical formulation. In the modern technological environment, the conventional methodology of design cannot compete with the modern trends of Computer Aided Engineering (CAE) techniques. The constant search for new innovative design in the engineering field is a common trend. To build highly optimized product, which is the basic requirement for survival in the global market today? All round efforts were put forward in this direction. Software professional and technologists have developed various design packages.

1.2 WORKINC OF FINITE ELEMENT METHOD

The finite element method is a numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. Although originally developed to study stresses in complex airframe structures. it has since been extended and applied to the broad field of continuum mechanics. Because of its diversity and flexibility as an analysis tool. it is receiving much attention in engineering schools and in industry.

(a) Finite difference

(b) Finite element Discretization of a turbine blade profile.



(a) **(b)**

Figure 1.1 Discretization of turbine blade profile

On the other hand, the finite element model (using the simplest two-dimensional element—the triangle) gives a better approximation to the region. Also, a better approximation to the boundary shape results because straight lines of any inclination represent the curved boundary. This example is not intended to suggest that finite element models are decidedly better than finite difference models for all problems. The only purpose of the example is to demonstrate that the finite element method is particularly well suited for problems with complex geometries. Still another numerical analysis method is the boundary element method (boundary integral equation method) this method uses Green's theorem to reduce the Dimensionality of the problem; a volume problem is reduced to a surface problem, a surface problem is reduced to a line problem. So far, the essence of the finite element method is explained. The next part would be to get into the details of the same. In a continuum problem of any dimension the field variable (whether it is pressure, temperature, displacement, stress, or some other quantity) possesses infinitely many values because it is a function of each generic point in the body or solution region. Consequently, the problem is one with an infinite number of unknowns. The finite element Discretization procedures reduce the problem to one of a finite number of unknowns by dividing the solution region into elements and by expressing the unknown field variable in terms of assumed approximating functions within each element. The approximating functions (sometimes called interpolation functions) are defined in terms of the values of the field variables at specified points called nodes or nodal points. Regardless of the approach used to find the element properties, the solution of a continuum problem by the finite element method always follows an orderly step-by-step process. To summarize in general terms how the finite clement method works we will succinctly list these steps now.

1. Discretize the Continuum: The first step is to divide the continuum or solution region into elements. In the example, the turbine blade has been divided into triangular elements that might be used to find the

temperature distribution or stress distribution in the blade. A variety of element shapes may be used, and different element shapes may be employed in the same solution region. Indeed, when analyzing an elastic structure that has different types of components such as plates and beams, it is not only desirable but also necessary to use different elements in the same solution.

2. Select Interpolation Functions: The next step is to assign nodes to each element and then choose the interpolation function to represent the variation of the field variable over the element. The field variable may be a scalar, a vector, or a higher-order tensor. Often, polynomials are selected as interpolation functions for the field variable because they are easy to integrate and differentiate. The degree of the polynomial chosen depends on the number of nodes assigned to the element, the nature and number of unknowns at each node, and certain continuity requirements imposed at the nodes and along the element boundaries. The magnitude of the field variable as well as the magnitude of its derivatives may be the unknowns at the nodes.

3. Find the Element Properties: Once the finite element model has been established (that is, once the elements and their interpolation functions have been selected), we are ready to determine the matrix equations expressing the properties of the individual elements. For this task we may use one of the three approaches just mentioned: the direct approach, the variation approach, or the weighted residuals approach.

4. Assemble the Element Properties to Obtain the System Equations: To find the properties of the overall system modeled by the network of elements we must "assemble" all the element properties. In other words, we combine the matrix equations expressing the behavior of the elements and form the matrix equations expressing the behavior of the entire system. The matrix equations for the system have the same form as the equations for an individual element except that they contain many more terms because they include all nodes. The basis for the assembly procedure stems from the fact that at a node, where elements are interconnected, the value of the field variable is the same for each element sharing that node. A unique feature of the finite element method is that assembly of the individual element equations generates the system equations.

5. Impose the Boundary Conditions: Before the system equations are ready for solution they must be modified to account for the boundary conditions of the problem. At this stage we impose known nodal values of the dependent variables or nodal loads.

6. Solve the System Equations: The assembly process gives a set of simultaneous equations that we solve to obtain the unknown nodal values of the problem. If the problem describes steady or equilibrium

behavior, then we must solve a set of linear or nonlinear algebraic equations.

7. Make Additional Computations (If desired): Many times we use the solution of the system equations to calculate other important parameters. For example, in a structural problem the nodal unknowns are displacement components. From these displacements we calculate element strains and stresses. Similarly, in a heat-conduction problem the nodal unknowns are temperatures, and from these we calculate element heat fluxes.

1.3. ENGINEERING APPLICATIONS OF THE FEM

Although the method has been extensively used in the field of structural mechanics, it has been successfully applied to solve several other types of engineering problem, such as heat conduction, fluid dynamics, seepage flow, and electric and magnetic fields. These applications prompted mathematicians to use this technique for the solution of complicated boundary value and other problems. In fact, it has been established that the method can be used for the numerical solution of ordinary and partial differential equations the general applicability of the finite element method can be observing.

1.4.1 INTRODUCTION TO FEA

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M. J. Turner, R. W. Clough, H, C, Martin, and L. 1. Top established a broader definition of numerical analysis.

The paper centered on the "stiffness and deflection of complex structures". FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design. and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

There are generally two types of analysis that are used in industry: 2D modeling, and 3D modeling.

While 2D modeling consent's simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling

schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation. and many also are capable of testing a material all the way to fracture.

FEA uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements. A wide range of objective functions (variables within the system) are available for minimization or maximization:

- Mass, volume, temperature
- Strain energy, stress strain
- Force, displacement, velocity, acceleration
- Synthetic (User defined)

There are multiple loading conditions which may be applied to a system. Some examples are shown:

- Point, pressure, thermal, gravity, and centrifugal static loads
- Thermal loads from solution of heat transfer analysis
- Enforced displacements
- Heat flux and convection
- Point, pressure and gravity dynamic loads

Each FEA program may come with an element library, or one is constructed over time. Some sample elements are:

- Rod elements

- **Beam elements**
- **Plate/Shell/Composite elements**
- **Shear panel**
- **Solid elements**
- **Spring elements**
- **Mass elements**
- **Rigid elements**
- **Viscous damping elements**

Many FEA programs also are equipped with the capability to use multiple materials within the structure such as:

- **Isotropic, identical throughout**
- **Orthotropic, identical at 90 degrees**
- **General anisotropic, different throughout**

TYPES OF ENGINEERING ANALYSIS

Structural analysis consists of linear and non-linear models. Linear models use simple parameters and assume that the material is not plastically deformed. Non-linear models consist of stressing the material past its elastic capabilities. The stresses in the material then vary with the amount of deformation as in.

Vibrational analysis is used to test a material against random vibrations, shock, and impact. Each of these incidences may act on the natural Vibrational frequency of the material which, in turn, may cause resonance and subsequent failure. Fatigue analysis helps designers to predict the life of a material or structure by showing the effects of cyclic loading on the specimen. Such analysis can show the areas where crack propagation is most likely to occur. Failure due to fatigue may also show the damage tolerance of the material.

Heat Transfer analysis models the conductivity or thermal fluid dynamics of the material or structure. This may consist of a steady-state or transient transfer. Steady-state transfer refers to constant thermal properties

in the material that yield linear heat diffusion.

RESULTS OF FINITE ELEMENT ANALYSIS

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested. In practice, a finite element analysis usually consists of three principal steps:

I. Preprocessing: The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical "preprocessor" to assist in this rather tedious chore. Some of these preprocessors can overlay a mesh on a preexisting CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

2. Analysis: The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations

$$K_{ij}u_j = f_i$$

Where u and f are the displacements and externally applied forces at the nodal points. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.

3. Post processing: In the earlier days of Finite Element Analysis, the user would pore through reams of numbers, generated by the code, listing displacements and stresses at discrete position within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. A typical postprocessor display overlays colored contours representing stress levels on the model, showing a full field picture similar to that of photo elastic or more experimental results.

SOLUTION

In the solution phase of the analysis, the computer takes over and solves the simultaneous equations that the finite element method generates. The results of the solution are- a) nodal degree of freedom values which form the primary solution and b) derived values, which form the element solution. The element solution is usually calculated at the element integration points. Several methods of solving the simultaneous equations are available in the ANSYS program me, frontal solution, sparse direction solution. Jacobi Conjugate Gradient solution, Precondition Conjugate Solution and an automatic iteration solver option. The frontal solver is the default.

SELECTING A SOLVER

The following table provides general guidelines you may find useful in selecting which solver to use for a given problem is shown in the table 4.1.

MODEL GENARATION

The ultimate purpose of a finite element analysis is to re-create mathematically the behavior of an actual engineering system. In other words, the analysis must be an accurate mathematical model of a physical prototype. In the broadest sense, this model comprises all the nodes, elements, material properties, real constants, boundary conditions and other features that are used to represent the physical system.

In ANSYS terminology, the term model generation usually takes on the narrower meaning of generating the nodes and elements that represents the spatial volume and connectivity of the actual system. The ANSYS program me offers you the following approaches to model generation.

- Creating a solid model within ANSYS.
- Direct generation Importing a model created in a computer aided design (CAD) system

1.4.2 INTRODUCTION TO ANSYS

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behavior of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations. ANSYS is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. ANSYS is also used in Civil and Electrical Engineering, as well as the Physics and Chemistry departments. ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping. With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of a design on the whole behaviour of the product, be it electromagnetic, thermal, mechanical etc.

GENERIC STEPS TO SOLVING ANY PROBLEM IN ANSYS

Like solving any problem analytically, you need to define

(1) Your solution domain.

(2) The Physical Model.

(3) Boundary Condition.

(4) The Physical Property.

You can solve the problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in the otherwise too complex situation. Below describes the process in the terminology slightly more attend to the software.

Build Geometry

Construct a two or three dimensional representation of the object to the model and tested using work plane co-ordinate system with in the ANSYS.

Define Material Properties

Now that the part exists define the library of the necessary material that compose the object (or projects) being modeled. This includes thermal and mechanical properties.

Generate Mesh

At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down in to the finite pieces.

Apply Loads

Once the system is fully designed, the last task is to burden the system with constraints, such as physical loading or boundary conditions.

Obtain Solution

This is actually a step, because ANSYS need to understand with in what state (stead-state, transient state etc..) the problem must be solved.

Present the Results

After the solution has been obtained, there are many ways to present ANSYS results, choose from any options such as Table, Graphs and Contour Plots.

LITERATURE REVIEW

By P.V. Krishnakanth, G Nara Raj, R D V. Prasad, R. Saisrinu [1]

Withstanding of gas turbine blades for the elongations is a major consideration in their design because they are subjected to high tangential, axial, centrifugal forces during their working conditions. Several methods have been suggested for the better enhancement of the mechanical properties of blades to withstand these extreme conditions, This Project summarizes the design and analysis of Gas turbine blade, on which CATIA v5 is used for design the solid model of the turbine blade With the help of the spline and extrude options, ANSYS 18.1 software is used analysis of FE model generated by meshing of the blade using the Solid brick element present in the ANSYS software itself and thereby applying the boundary condition. This project specifies how the program makes effective use of the ANSYS pre-processor to analyze the complex turbine blade geometries and apply boundary conditions to examine SteadyState thermal & structural performance of the blade for N 155, Haste alloy X, Inconel 625 materials. Finally stating the best suited material among the therefrom the report generated after analysis. From this the results are stated and reported

Sagar P.Kauthalkar, Mr.Devendra S. Shikarwar, Dr. Pushpendra Kumar Sharma [2013] [2]

The purpose of turbine technology is to extract maximum quantity of energy from the working fluid to convert it into useful work with maximum efficiency. That means the Gas Turbine having maximum reliability, minimum cost, minimum supervision and minimum starting time. The gas turbine obtains its power by utilizing the energy of burnt gases and the air. This is at high temperature and pressure by expending through the several rings of fixed and moving blades. A high pressure of order 4 to 10 bar of working fluid which is essential for expansion, a compressor is required. The quantity of working fluid and speed required are more so generally a centrifugal or axial compressor is required. The turbine drives the compressor so it is Coupled to the turbine shaft. After compression the working fluid are to be expanded in a turbine. Then assuming that there were no losses in either component, the power developed by the turbine can be increased by increasing the volume of working fluid at constant pressure or alternatively increasing the pressure at Constant volume. Either of these may be done by adding heat so that the ten1Perature of the working fluid is increased after compression. To get a higher temperature of the working fluid a combustion chamber is required where combustion of air and fuel takes Start 1.5 spacing from here place giving temperature rise to the working fluid.

G.Narendranath & S.Suresh [3]

In the present work, the first stage rotor blade off the gas turbine has been 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 CAT1A V5R1 5. 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 is meshed in HYPERMESH 7.0, meshing software. The thermal boundary condition such as convection and operating temperatures on the rotor blade is obtained by theoretical modeling. Analytical approach is used to estimate the tangential, radial and centrifugal forces. The results obtained are discussed and reported.

V.Veeraragavan [2012] [4]

In this research paper is mainly apprehensive with aircraft Gas Turbine Engine. Turbine blade is an important part of gas turbine engine. The research focus of 10 C4/60 C 50 turbine blade model, because of its common use in all types of aircraft engines. Investigate used, Pro-e model and ANSYS tools. Present research was focused on using Finite element methods (FEM) to predict the location of possible temperature areas on turbine blades. The conventional alloys such as titanium, zirconium, molybdenum, super alloys are chosen for analysis. Initially the model is created with the help of Pro-e and then it is imported to Ansys. The static analysis of solid model is carried out by applying temperature from external circumference tip of turbine blade to root of the blade and the temperature distribution is plotted. At that time measured the maximum temperature withstood capacity in gas turbine blade. Finally, the entire four alloy materials are compared with respect to temperature distribution to found out of the best one. Then suggested to which material is better performing in gas turbine engine applications.

Theju V, Uday P S, PLV Gopinath Reddy, C.J.Manjunath[2012] [5]

The objective of this project is to design and stresses analyze a turbine blade of a jet engine. An investigation for the usage of new materials is required. In the present work turbine blade was designed with two different materials named as Inconel 718 and Titanium T-6. An attempt has been made to investigate the effect of temperature and induced stresses on the turbine blade. A thermal analysis has been

carried out to investigate the direction of the temperature flow which is been develops due to the thermal loading. A structural analysis has been carried out to investigate the stresses, shear stress and displacements of the turbine blade which is been develop due to the coupling effect of thermal and centrifugal loads. An attempt is also made to suggest the best material for a turbine blade by comparing the results obtained for two different materials (Inconel 718 and titanium T6). Based on the plots and results Inconel 718 can be consider as the best material which is economical, as well as it has good material properties at higher temperature as compared to that of Titanium T6.

V Prithvi Raj, K. Arun Kumar [2014] [6]

Gas Turbines are essential in generation of power in the field of aviation etc. proper design of all the elements of the gas turbine will play a pivotal role in providing an efficient and ergonomic gas turbine. Gas turbine rotor is one of the key elements of gas turbine. Hence analysis of gas turbine rotor is essential. Gas turbine rotor assembly is mainly subjected to centrifugal stresses with high temperature gradients. If these stresses are beyond the threshold limit of the strength of the material of the gas turbine rotor, rotor failure will occur. In the present work, the gas turbine rotor assembly is analyzed for thermal loads, centrifugal forces and natural frequency due to the mass of the rotor assembly. The gas turbine rotor will be analyzed as a segment of the design and modeling of gas turbine. In this work structural and thermal characteristics in gas turbine rotor assembly due to various operating conditions will be analyzed by varying the suitable materials, analysis using FEA software ANSYS workbench-14.5 and results are presented. From the results presented, one can say that the structural and thermal characteristics of the rotor will be reduced to eliminate the use of high level materials.

Ganta Nagaraju I, Venkata Ramesh Mamilla, M.V. Mallikarjun [2013] [7]

This paper studies about the 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 turbine blade geometries and apply boundary conditions. The objective of this paper is to get the natural frequencies and mode shape of the turbine blade.

Gujadel and M B Bhambere [2014] [8]

The Gas Turbine obtains its Power by utilizing the energy of burnt gases and the Air which is at high temperature and pressure by expanding through the several rings of fixed and moving blades. Since the turbine blades are working at high temperature and pressure there are extreme stresses developed on turbine blades. The first centrifugal stresses act on the blade due to high angular speeds, and second is thermal stresses that arise due to temperature gradient within the blade material. The present paper is review of various analyses done on turbine blades and there are various factors effects on turbine blade. This paper will be helpful for those who are working in the area of power plants. [19]

Josin George [2014] [9]

Gas turbine blades will subject to high tangential, axial and centrifugal forces during their working conditions. While withstanding these forces gas turbine blades may have subjected to elongation. Several methods have been suggested for the better enhancement of the mechanical properties of blade to withstand these extreme conditions. This project summarizes the design, analysis and modification of the cooling passage in the gas turbine blade design. On which CATIA V5 is used for design of solid model of the turbine blade with the help of the spline and extrude options. ANSYS 14.0 Software is used to analysis of finite element model generated by meshing of the blade by applying boundary conditions. From the analysis results the better material for first stage turbine blade is stated. After that by using the better material properties the cooling passage of the turbine blade is modified into serpentine model and changing the number of holes. [20]

V.Raga Deepu & RP.Kumar [2012] [10]

Turbine is a device designed to convert the heat energy of fuel into useful work such as mechanical shaft power. The gas turbine in its most common from is a rotary heat engine operating by means of series of processes consisting of air taken from the atmosphere increase of gas temperature by constant pressure combustion of the fuel the whole process being continuous. Turbine Blades are the most important components in a gas turbine power plant. A blade can be defined as the medium of transfer of energy from the to the turbine rotor. The turbine blades are mainly affected due to static loads. Also the temperature has significant effect on the blades. Therefore, the coupled (static and thermal) analysis of turbine blades is carried out using finite element analysis software ANSYS. It was observed that in the preliminary design, the rotor blades after being designed were analyzed only for the mechanical stresses but no evaluation of thermal stress was carried out.

B. Deepanraj, P. Lawrence and G. Sankaranarayanan [2014] [11]

Gas turbine is an important functional part of many applications. Cooling of blades has been a major concern since they are in a high temperature environment. Various techniques have been proposed for the cooling of blades and one such technique is to have axial holes along the blade span. Finite element analysis is used to analyze thermal and structural performance due to the loading condition, with material properties of Titanium- Aluminum Alloy. Six different models with different number of holes (7, 8, 9, 10, 11, and 12) were analyzed in this paper to find out the optimum number of holes for good performance. In Finite element analysis, first thermal analysis followed by structural analysis is carried out. Graphs are plotted for temperature distribution for existing design (12 holes) and for 8 holes against time. 2D and 3D model of the, blade with cooling passages are shown. Using ANSYS, bending stress, deflection, temperature distribution for number of holes are analyzed. It is found that when the numbers of holes are increased in the blade, the temperature distribution falls down. For the blade configuration with 8 holes, the temperature near to the required value i.e., 800°C is obtained. Thus a turbine blade with 8 holes' configuration is found to be the optimum solution.

Kamlesh Bachkar, W.S. Rathod [2014] [12]

This paper borders around analysis of turbine blade. The blade is a rotating part which converts kinetic energy into mechanical energy. Turbine blade is critical part of turbocharger which has shown increasing growth of failure damaging turbine disk. It deals with Static and thermal analysis of turbine blade which is made up of INCONEL 718 to estimate its performance. The causes of failure for turbine blade have also been found out. The investigation has been done using Solid Works 2012 and ANSYS 10.0 software. Solid Works 2012 is used for modeling of turbine blade and analysis has been done by ANSYS 14 Software. An attempt has been made to investigate the effect of induced stresses, pressure and temperature on the turbine blade. A structural analysis has been carried out to investigate the stresses and displacements of the turbine blade. A thermal analysis has been carried out to investigate the thermal gradient and thermal stress.

V. NagaBhushana Rao, L N. Niranjan Kumar, N. Madhulata and A. Abhijeet [2014] [13]

Turbine blades of a gas turbine are responsible for extracting energy from the high temperature, high pressure gases. These blades are operated at elevated temperatures in aggressive environments and are subjected to large centrifugal forces. As many as 42 percent of the failures in gas turbine engines were only

due to blading problems and the failures in these turbine blades can have dramatic effect on the safety and performance of the gas turbine engine. In this research paper, an attempt has been made to analyze the failure of gas turbine blade through Mechanical analysis. The blade under investigation belongs to a 30 MW gas turbine engines used in marine applications and is made of Nickel-Base super alloys. Before failure, the turbine blade was operated for about 10000 hours while its service life was expected to be around 15000 hours. Mechanical analysis has been carried out assuming that there might be failure in the blade material due to blade operation at elevated temperature and subjected to large centrifugal forces. The gas turbine blade model profile is generated by using CATIA V5 R21 software. The turbine blade is analyzed for its thermal as well as structural performance. It was observed that there was no evidence of rubbing marks on the tip section of turbine blade indicating the elongation of the blade is within the safe limit. Maximum stresses and strains are observed near to the root of the turbine blade and upper surface along the blade roots. Maximum temperatures are observed at the blade tip sections and minimum temperature at the root of the blade. Temperature distribution is decreasing from the tip to the root of the blade section. The temperatures observed are below w the melting temperature of blade material.

2.2. SCOPE OF THE PROJECT

Due to development of computers and subsequent deployment of numerical methods, it is now possible to model the components, simulate the conditions and perform testing on computer without actual model making, one of the most popular numerical methods used is the Finite Element (FEM) offered by the existing CAD/CAM/ CAL 'the most popular software based on finite Element Analysis is -ANSYS package', which is used in this work. The stress analysis in the fields of civil, mechanical and aerospace engineering, nuclear engineering is invariably complex and for many of the problems it is extremely difficult and tedious to obtain analytical solutions. In these situations, engineers usually resort to numerical methods to solve the problems. With the advent of computers, one of the most powerful techniques that has been developed in the engineering analysis is the finite element method and tile method being used for the analysis of structures/solids of complex shapes and complicated boundary conditions. In the present study, the First stage rotor blade of the gas turbine has been analyzed using ANSYS 18.1 for Structural, Thermal and Modal Analysis. The material used in this study are three materials namely Titanium Alloy, Inconel 718 and Structural Steel.

CHAPTER 3:

GAS TURBINE

3.1 ANALYSIS OF STATIC AND DYNAMIC BEHAVIOR OF ROTOR BLADE

Analysis of static and dynamic behavior of the aero elasticity of turbo machine blade rotor blade is a basic problem in the aero elasticity of the turbo machines blades. The heart of these new machines is made by the blades and vanes which are subjected during operation due to very high thermal and mechanical stresses (combined effect of centrifugal force and thermal gradient) in aggressive environment the blades suffer during operation of several damages which limit the component overall life, life cycle fatigue, hot corrosion .Turbo machine rotor blades are subjected to different types of loading such as fluid or gas forces, inertia loads and centrifugal forces. Due to these forces various stresses are induced in rotor blades. So stress and strain mapping on a rotor blade provide a vital information concerning the turbo machine design and lead to the detection of critical blade section. The present paper deals with the stress analysis of a typical blade made up of nickel super alloy, which is subjected to centrifugal loading. The analysis results show that stress is sever due to centrifugal forces compared that due to dynamic gas forces. Here in this case the effect of thickness, twist and taper of the blade was considered at the root of the blade here generally failure is occurring. The various blade shapes viz. rectangular, airfoils with some angle twist, taper airfoil are taken into consideration. in this paper linear static analysis for determining von misses' stresses, deformation in Z direction was determined using Finite element analysis software.

Aero engine turbine and compressor blades operate at speed range 5000 to 15000 r.p.m. with temperature ranging from 50 to 900 degree centigrade. Hence depending on the stage of operation, Wading material is usually an Al alloy, stainless steels, titanium alloys and nickel-based alloys. The tolerances on the blades are usually in the range of 0.05 mm to 0.15 mm on the airfoil.

The blades have a complex airfoil structure and with varying airfoil shape at different sections along the length of blade. There is always a twist in the airfoil sometimes of the order 60 degrees. These complex configurations are required as the gases are to be smoothly guided along the different stages of the compressor and turbine without turbulence to achieve maximum thrust from the engine.

Fatigue failure results from a combination of steady stress, vibratory stress, and material imperfections. However, the size of microscopic imperfections is difficult to control. Hence, stress-range diagrams are used to quantify the allowable vibratory stress amplitudes to avoid fatigue damage. Advanced turbo machinery blading is designed to have high steady stress levels.

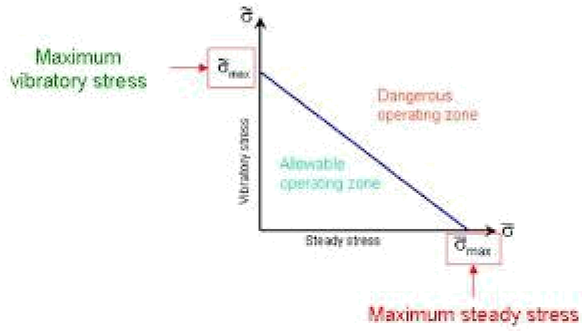


Figure 3.1 - Stress Range Diagram

Thus high cycle fatigue occurs because of high mean stress low amplitude vibratory loading of aerofoils.as shown. It is initiated by the formation of small, often microscopic cracks.

It is essential to incorporate the Computer Aided Engineering in the turbocharger development and design process. Structural analysis of their component parts has been made so far mainly on the automotive and marine turbochargers, centrifugal compressors. We have made analysis of the stress red C by external force and pressure, analysis of thermal stress caused by heat turbine impellers and rotors as rotary parts. We have started establish the method of getting the optimum profile using stress value and displacement as limiting condition. This paper introduces the transition of the structural analysis of the impeller as the most important component of the turbocharger. introduced that products can gain the performance and reliability by using Computer Aided Engineering and shape optimization. The conventional method of impeller structural analysis has been the repeated work. The profile model is created and then analysis is made. The profile is changed according 10 the evaluation results. Then calculation is made again. In future, however it is important to adopt the process of getting the optimum profile where stress and displacements are used as limiting conditions. For the impeller, the optimum profiles of the blade and disk can be enumerated as such profiles. To cope with a big change in flow of product development from the 2 D to 3 D designing, the role of various types of Computer Aided Engineering is very important.

It is ideal that Computer Aided Engineering be incorporated in the development design process and used in the initial phase of development designing. To achieve this, we will make more efforts to solve many issues and to make contribution to further development of the subsequent Computer Aided Engineering and renovations in the true sense of manufacturing. In this paper, blade operating and service conditions are taken into Consideration. The factors are inspection results, coating stripping, and cracks due to thermo-mechanical stresses during operation likely to be located on airfoil and on fillet radius, mechanical deformation modifications of design profile mainly on trailing edge, corrosion, and erosion, coating and

surface degradation due to high temperature on airfoil.

Therefore, in term of maintained equipment, manufacturing difficulties and costs, the blades are the most critical item of the gas turbines. These components are manufactured by some nickel and cobalt-based alloy able to Failure analysis, chemical and metallurgical analysis, thermal and stress to high temperature and mechanical stresses have undertaken several improvements aimed to increases the overall performance and the useful life. Fatigue analysis, chemical and metallurgical analysis, thermal and stress analysis, design review and proposed solution is taken into consideration's purpose of this upgrading package is to move the turbines first stage blades and vanes as much as close to the level of the other turbine stages which have achieved an outstanding performance in terms of reliability and scrap rate. This Paper compares the accuracy of elastic and elastoplastic solid continuum finite element analyses modelled with all hexahedral or all tetrahedral meshes. Eigenvalues of element stiffness matrices, linear static displacements and stresses, dynamic modal frequencies and plastic flow values are compared. Elements with both linear and quadratic functions are evaluated. A simple bar with a rectangular cross-section fixed at one end is modelled and results are compared to known analytical solutions wherever possible. The evaluation substantiates a strong preface for linear displacement hexagonal finite elements when compared to linear tetrahedral finite elements. The use of quadratic displacements formulated finite elements improves the performance of tetrahedral as all as hexahedral elements. The nonlinear elastoplastic comparison indicates that linear hexahedral elements may be superior even to quadratic tetrahedrons. In this paper, Eigenvalues of a square geometrical volume meshed with a single hexahedron is compared to the same geometrical volume meshed with five tetrahedrons. Next, results of a linear elastic, fixed end bar meshed with either all hexahedrons or all tetrahedrons are compared. Both bending and torsional results are considered. The computed vibration modes of the fixed end bar problem arc then evaluated. Finally, elastoplastic calculations of the fixed end bar vain meshed with both types of elements are evaluated. Numerous calculations have been conducted in this paper that compares the accuracy of all tetrahedral meshes to all hexahedral meshes First it was shown that the stiffness matrix Eigenvalues for linear tetrahedrons more generally large that those for linear hexahedrons.

3.2 DEVELOP THE ANALYSIS OF GAS TURBINE ROTOR BLADE:

This enables the designers to develop the analysis of gas turbine rotor blade more effectively and easily

a) To determine thermal stresses due to the high-temperature gradient.

b) To determine maximum stress induced in blades.

c) To determine the temperature distribution along the blade profile.

d) To maintain temperature and stresses within limits.

e) To determine the parameters influencing the stress concentration in rotor blades.

f) To determine the effect of change of material properties

3.3 INTRODUCTION TO GAS TURBINES

TURBO MACHINES:

Turbines, compressors and fans are all members of the same family of machines called turbo machines. Turbo machine is a power generating machine, which employs the dynamic action of a rotating element, the rotor. The action of the rotor changes, the energy level continuously, of the fluid flowing through the turbo machine.

The majority of turbo machines run at comparatively higher speeds. These machines are adiabatic machines with high fluid velocities

The power generating turbo machines decrease the energy level of the working fluids passing through them. The principle element in a turbo machine is the rotor, which performs the basic function of transfer of energy. The example of these type machines are Gas turbines, Steam Turbines, etc.

GAS TURBINE:

A gas turbine is an engine designed to convert the energy of a fuel into some form of useful power such as mechanical power or high - speed thrust of a jet. A Gas turbine basically consists of a gas generator section and a power conversion section.

GAS TURBINE THEORY

A simple gas turbine is comprised of three main sections a compressor, a combustor, and a turbine. The Gas Turbine operates on the principle of the Brayton cycle, where compressed air mixed with fuel, and burned under constant pressure conditions. The resulting hot gas is allowed to expand through a turbine to perform work. In a 33% efficient gas turbine approximately two / thirds of the work is spent compressing the air, the rest is available for other work i.e(mechanical drive, electrical generation).

A Brayton cycle is characterized by two very significant parameters: Pressure ratio and Firing temperature.

The thermodynamic cycle efficiency and power output of a gas turbine depends on these two factors

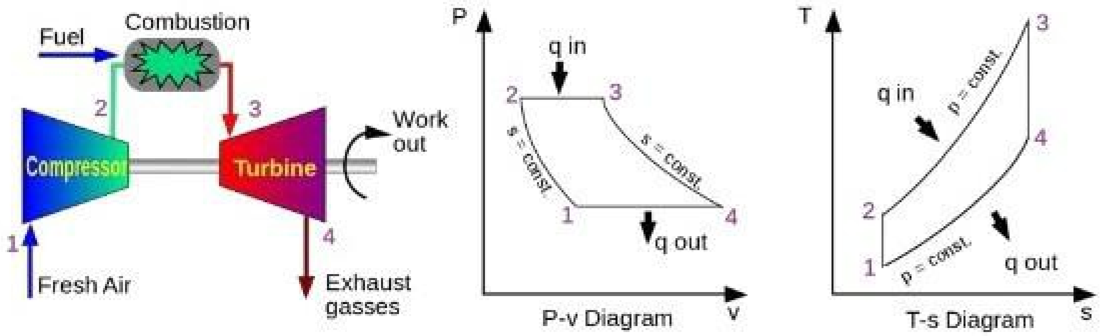


Fig 3.2 Working of Gas Turbines(Cycle)

GAS TURBINE DESCRIPTION

The main sections of Gas turbine are:

(1) **Compressor section**

(2) **Combustion section**

(3) **Turbine section**

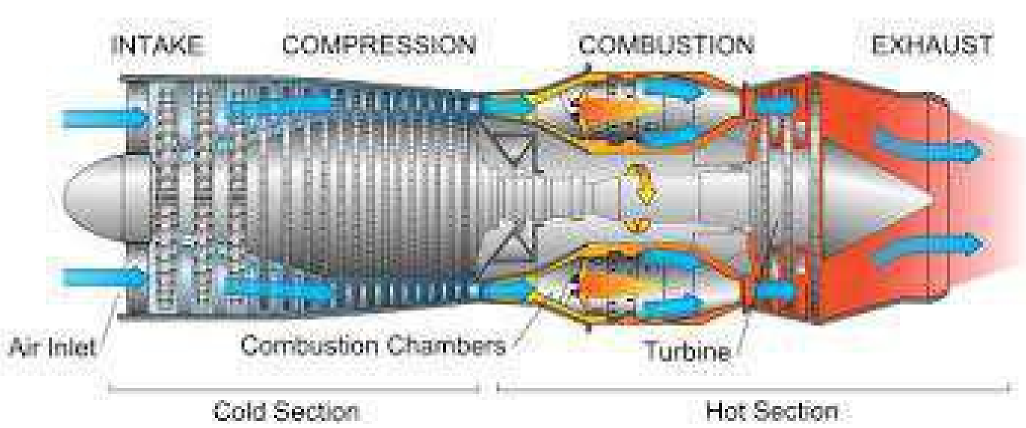


Fig. 3.3 Gas Turbine Outline

1. COMPRESSOR SECTION

The axial flow compressor section consists of the compressor rotor and the enclosing casing. Included with the compressor casing are the inlet guide vanes, are the 17 stages of the rotor and stator blading and the 2 exit guide vanes.

In the compressor, air is confined to the space between the rotor and stator blading where it is compressed in stages by series of alternate rotating and stationary airfoil shaped blades. Rotor blades supply the force needed to compress the air in each stage and the stator blade guide the air so that it enters the following rotor stage at the proper angle. The compressed air exits through the compressor discharge casing to the combustion chamber. Air is extracted from the compressor for turbine cooling, for bearing sealing & during start up for pulsation control.

Since minimum clearance between rotor & stator provides best performance in a compressor, parts have to be made & assembled very accurately.

COMPRESSOR ROTOR:

The axial - flow compressor rotor assembly consists of 16 blade and wheel assemblies and one blade and stub shaft assembly. The blade and stub shaft assembly and the blade wheel assemblies are rabbeted and bolted together concentrically around the rotor axis. The bolt holes are countersunk in the stub shaft. This machining keeps the bolt heads and nuts flush with the wheel face and reduces windage loss.

The stub shaft is machined to provide the forward and aft thrust faces and the journal for the No1 bearing assembly, and the sealing surfaces for the No1 bearing oil seals and the compressor low pressure air seal.

The compressor assembly rotor consists of the following:

1. **A forward stub shaft on which are mounted their 1st stage blades.**
2. **Sixteen blade and wheel assembly (rotor stages 2 to 17 inclusive).**

The compressor rotor assembly is dynamically balanced before it is assembled to the prebalanced turbine rotor assembly. This completed assembly is then dynamically balanced. The balance corrections are carefully and properly distributed so as to compensate for internal bending moments in the complete assembly.

COMPRESSOR STATOR:

The compressor casing (Stator) encloses the compressor portion of the rotor and is divided into four sections:

-Inlet Casing

-Forward Casing

-Aft Casing

-Discharge Casing:

All of these sections are split horizontally to facilitate servicing.

Inlet Casing:

The inlet section directs the flow of outside air from the air inlet equipment into the compressor blading. This section contains the variable inlet guide vane assembly, No 1 bearing assembly, and the low pressure air seals.

Forward Casing:

The forward section of the compressor casing is downstream of the inlet section. It contains the stator blading for stages 0 through 3. Bleed air from the 4th rotor stage (Between the 3rd and 4th stator stages) can be extracted through four ports which are located about the aft section of the compressor casing.

Aft Casing:

The aft section, downstream of the forward section, contains the stator blading for stages 4 through 9. Bleed air from the 10th rotor stage (Between the 9th and 10th stator stages) can be extracted through four ports which are located in radial alignment with the ports used for 4th stage air extraction.

Discharge Casing:

The discharge section of the compressor casing, downstream of the aft section, contains the stator blading for the stages 10 through 16, and exit guide vane stages 1 and 2. A radially enlarged (Bulkhead) portion of this section provides the mounting surface for the combustion chambers. Ten air foil shaped

support struts are secured equidistantly about the aft surface of the bulkhead and angle inward to support the inner casing assembly (inner barrel). The space between the forward portion of the inner barrel and the discharge section outer shell, forms an annular air path that the high pressure air passes through to enter the combustion section. This area is designed to decelerate the air flow and increase static pressure of the combustion air supply.

Blading:

The stator blades have dovetail-shaped bases that fit into dovetail shaped openings into the two-piece, semicircular ring. The ring fits into the groove of the same shape machined in the compressor casing wall. Locking keys prevent the rotating of the blade rings. The rotor blades also have dovetailed bases of a wide angle design which fit into the matching dovetail openings in the wheels. The rotor blades are peened in place.

Variable Inlet Guide Vanes:

The variable inlet guide vanes (in conjunction with 10th stage air extraction) permits fast, smooth acceleration of the turbine without compressor surge (pulsation). A hydraulic cylinder, mounted on a base cross member, actuates the inlet guide vanes through a large ring gear and multiple small pinion gears.

2. COMBUSTION SECTION

The combustion section consists of combustion chambers, fuel nozzles, flame detection equipment, spark plugs, and transition pieces.

The combustion chambers are arranged concentrically around the axial-flow compressor and are bolted to the compressor discharge section bulk head. Air for combustion is supplied directly from the axial flow compressor to the combustion chambers. Fuel is fed into the chambers through fuel nozzles that extend into each chambers liner cap.

Combustion chambers:

The high pressure air flow from the compressor discharges into the annular space created by the aft end of the discharge casing, frame assembly and the forward section of the turbine shell. Up to this point, the air flow has been in aft direction; now the air flow reverses. The air enters the combustion chambers and

flows forward, entering the liner through holes and louvers in the liner wall. A portion of the air reaches the head of the combustion chamber and enters the liner cap and the vortex generator nozzle.

The air flow through the combustion chambers has three functions

To oxidize the fuel

To cool the metal parts

And to adjust the extremely hot combustion products to the desired turbine inlet temperature

Spark plugs:

Combustion of fuel and air mixture is initiated by the spark plugs with retracting electrodes. The spark plugs, installed in two of the combustion chambers, receive power from the ignition transformers. The chambers without the spark plugs are fired with flame from the fired chambers through interconnecting cross fire tubes.

Ultra violet flame detector:

During the starting sequence, it is essential that an indication of the presence or absence of flame be transmitted to the control system. For this reason, a flame monitoring system is used consisting of two sensors which are installed on two adjacent combustion chambers and an electronic amplifier which is mounted in the turbine control panel.

Fuel nozzles:

Each combustion chamber is equipped with a fuel nozzle that emits the metered amount of fuel into the combustion liner. Gaseous fuel is admitted directly into each chamber through metered holes located at the outer edge of the swirl plate. When the liquid fuel is used it is atomized in the nozzle swirl chamber by means of higher pressure air. The atomized fuel/air mixture is then sprayed into the combustion zone. Action of swirl tip imparts a swirl to the combustion air with the result of more complete combustion and essentially smoke free operation of the unit.

Crossfire tubes:

The 10 combustion chambers are interconnected by means of crossfire tubes. These tubes enable flame from the fired chambers containing spark plugs to propagate to the unfired chambers.

Transition pieces:

The transition pieces are the hot gases path link between the combustion chambers and the first stage nozzle. They are clamped to the forward side of the nozzle assembly. The nozzle assembly is sealed at both its outer and inner periphery to prevent the leakage of hot gases. Before the compressor discharge air flows into the combustion chamber, it must first pass around the transition pieces. This encounter affords an exchange of heat i.e. cooling the transition pieces and preheating the combustion air.

3. TURBINE SECTION

In the turbine section, high temperature gases from the combustion section are converted to the shaft horsepower. The power required to drive the load package and the compressor is provided by the two stage turbine rotor in case of Fr5 design and by three stage turbine in case of Fr6 design. In case of Fr5 design the first and second stage wheels are bolted together to make up a single unit through which first and second stage nozzles direct the flow of combustion gases.

Turbine rotor assembly

The turbine rotor assembly consists of the distance piece and the first and second stage turbine wheels and buckets in case of Fr5 design.

The turbine wheels are forged of high temperature alloy steel. The second stage wheel is forged with a stub shaft on which the journal and sealing surfaces is machined for the No 2 bearing and its oil seal. At the stub shaft end is a flange to couple the shaft to the driven device. The buckets have “pine tree “slots.

The individual components of the rotor assembly are pre-balanced and assembled so that the complete rotor assembly will required a minimum of correction. The rotor assembly is dynamically balanced with any required corrections carefully distributed to compensate for internal bending moments.

The turbine rotor assembly is bolted to the pre-balanced compressor rotor assembly This complete rotor assembly is again dynamically balanced with any required corrections carefully distributed to compensate for internal bending moments.

3.4 FUNCTIONAL DESCRIPTION OF GAS TURBINE

For most of the Gas Turbine (G.T.)'s has single shaft, simple cycle, heavy duty G.T. unit driving a Generator. FUEL & Air are used by G.T. unit to produce the shaft horse power necessary to drive certain accessories & ultimately the driven load Generator.

The G.T. unit comprises of starting device, support system, axial flow compressor, combustion system components, a three stage turbine are directly connected with an in line, single shaft rotor supported by two pressure lubricated bearings. The inlet end of the rotor shaft is coupled to an accessory gear having integral shafts that drive the fuel pump, lubrication pump & other system components.

G.T. functional description:

When the turbine starting system is actuated & the clutch is engaged, ambient air is drawn through the inlet plenum assembly, filtered, then compressed in the 17th stage, axial flow compressor. For the pulsation protection during start up, the inlet guide vane (IGV)'s are open & the variable IGV's are in the closed position.

When the speed relay corresponding to 95% speed actuates, the 11th stage extraction bleed valves close automatically & the variable IGV actuator energizes to open the IGV to the normal turbine operating position.

Compressed air from the compressor flows into the annular space surrounding the 14 combustion chambers, from which it flows into the space between the outer combustion castings & the combustion liners. The fuel nozzles introduce the fuel into each of the 14 combustion chambers where it mixes with combustion air & is lighted by both (or one, which is sufficient) of the two spark plugs. At the instant one or both of the two spark plugs equipped combustion chambers is ignited, the remaining combustion chambers are also ignited by cross fire tubes that connect the reaction zones of the combustion chambers. After the turbine rotor approximates operating speed, combustion chambers pressure causes spark plugs to retract to remove their electrodes from the hot flame zone.

The hot gases from the combustion chambers expand into the 14 separator transition pieces attached to the aft end of the combustion liners & flow towards the 3 stage turbine section of the machine. Each stage consists of a row of fixed nozzles followed by a row of rotatable turbine buckets. In each nozzle row, the kinetic energy of the jet is increased, with an associated pressure drop & in each following row of moving buckets, a portion of the KE of the jet is absorbed as useful work on the turbine rotor.

After passing through the 3rd stage buckets, the exhaust gases are directed into the exhaust hood & diffuser which contains a series of turning vanes to turn the gas from the axial direction to a radial direction, thereby minimizing exhaust gases into the exhaust plenum. The result shaft rotation is used to turn the generator rotor & drive certain accessories.

3.5 APPLICATIONS OF GAS TURBINES:

Gas Turbines are used in various fields for various applications such as:

POWER GENERATION

Higher capacity gas turbines are widely used in power generation and stand by

The grid system: Power is produced and distributed continuously to large distances through a grid system

Stand by generation: Power is produced for emergency uses in hospitals, public building only for local purpose.

MECHANICAL DRIVE APPLICATION

To pump gas and oil through pipeline.

Instead of engines where power ranger is of range above 6 MW where the scope of diesel engines is almost nil.

AUTOMOTIVE APPLICATION

For high speed engines, For gas cars, Hybrid electric vehicles. (The gas turbines supply power to recharge batteries of electric motors), For racing cars, In battle tanks.

MARINE APPLICATION

Merchant containers, Submarines.

AIR CRAFT APPLICATION

Unmanned vehicle systems, Commercial aircrafts and military trainee, Advanced military fighters, Missiles.

3.6 THEORY OF OPERATION

The working fluid contains Potential energy (pressure head) and kinetic energy (velocity head). The fluid may be compressible or incompressible. Several principles are employed by turbines to collect this energy.

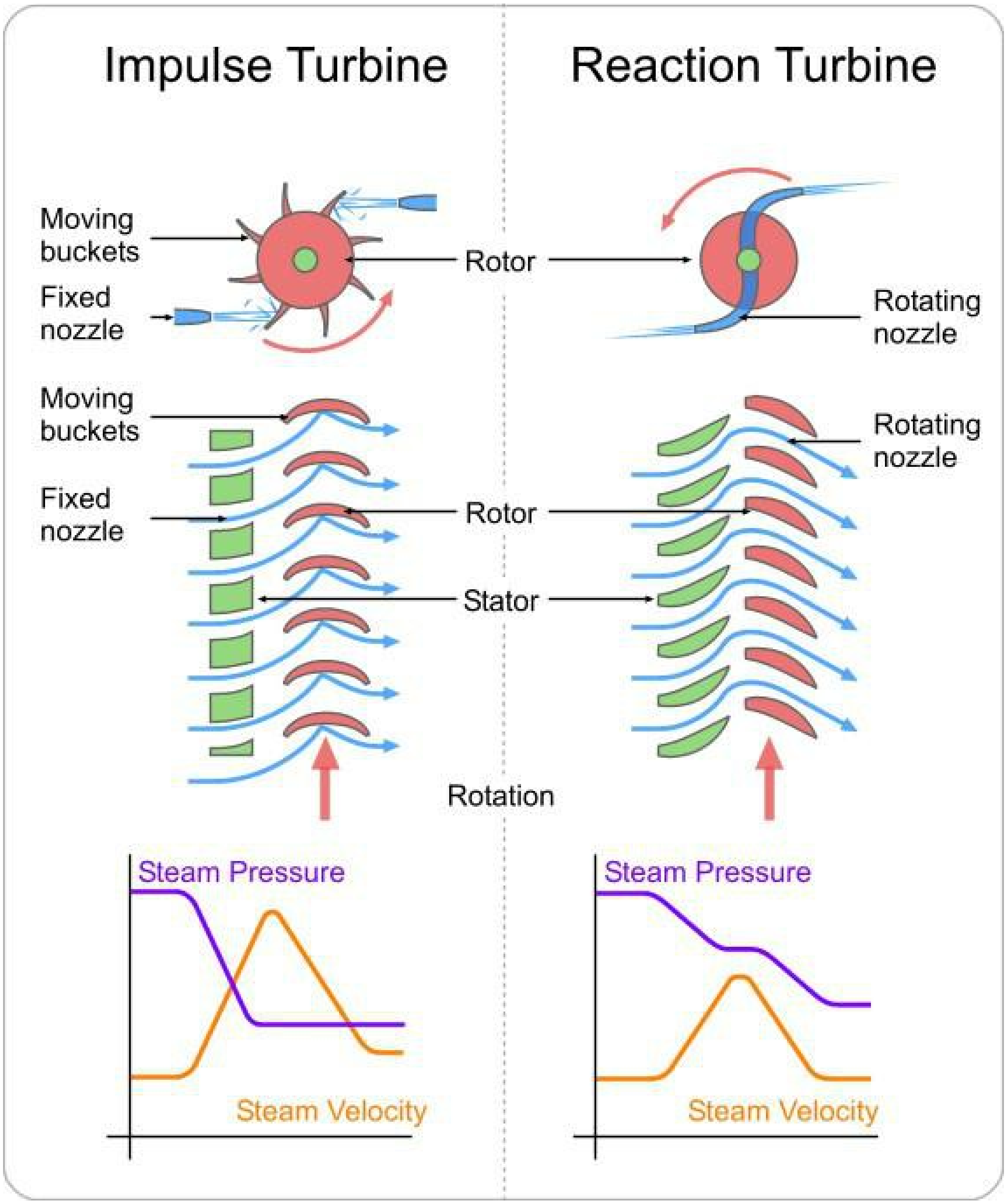


Fig. 3.4 Impulse and Reaction Turbine

Impulse turbines

Change the direction of flow of a high-velocity fluid or gas jet. The resulting of impulse spins the turbine and leaves the fluid flow with diminished kinetic energy. There is no pressure change of the fluid or gas in the turbine blades (the moving blades), as in the case of a steam or gas turbine; the entire pressure drop takes place in the stationary blades (the nozzles). Before reaching the turbine, the fluid's pressure head is changed to velocity head by accelerating the fluid with a nozzle. Peloton wheels and de Laval turbines use this process exclusively. Impulse turbines do not require a pressure casement around the rotor since the fluid jet is created by the nozzle prior to reaching the blading on the rotor. Newton's second law describes the transfer of energy for impulse turbines.

Reaction turbines

Develop torque by reacting to the gas or fluids pressure or mass. The pressure of the gas or fluid changes as it passes through rotor blades. A pressure casement is needed to contain the working fluid as it acts on the turbine stage(s) or the turbine must be fully immersed in the fluid flow (such as with wind turbines). The casing contains and directs the working fluid and, for water turbine maintains the suction imparted by the draft tubes, Francis turbines and most steam turbines use this concept. For compressible working fluids, multiple turbine stages are usually used to harness the expanding gas efficiently. Newton's third law describes the transfer of energy for reaction turbines. In practice, modern turbine designs use both reaction and impulse concepts to varying degrees whenever possible. Wind turbines use an airfoil to generate a reaction lift from the moving fluid and impart it to the rotor. Wind turbines also gain some energy from the impulse of the wind, by deflecting it at an angle. Cross-flow turbines are designed as an impulse machine, with a nozzle, but in the low head, applications maintain some efficiency through reaction, like a traditional water wheel. Turbines with multiple stages may utilize either reaction or impulse blading at high pressure. Steam turbines were traditionally more impulse but continue to move towards reaction designs similar to those used in gas turbines. At low pressure, the operating fluid medium expands in volume for small reductions in pressure. Under these conditions, blading becomes strictly a reaction type design with the base of the blade solely impulse. The reason is due to the effect of the rotation speed for each blade. As the volume increases, the blade height increases, and the base of the blade spins at a slower speed relative to the tip. This change in speed forces a designer to change from impulse at the base, to a high reaction style tip.

3.7. BLADE MATERIALS

Proper selection of blade material plays an important role in blade design. The factors that influence the selection of blade materials are:

- **Method of manufacture.**
- **Ease of machining.**
- **Ability to produce blade sections free from flaws.**
- **Ductility both allow for rolling of shapes.**
- **Capacity for being welded.**
- **Ease of forging.**
- **Operating Conditions.**
- **Suitable tensile strength at high temperature.**
- **Resistance to creep.**
- **Cost.**

The commonly used blade materials are:

Brass

Brass (70 to 72% Cu and 28 to 30% Zn) is suitable for temperature up 230ºC. This is rarely used nowadays.

Copper Nickel

This is an alloy containing about 80 % Cu, 19 % Ni and fraction of iron and magnesium.

Nickel Brass

It is suitable for temperature range up to 230ºC and contains 50 % Cu, 10 Zn and 10 Ni. It may be cold drawn.

Manganese copper

Its composition is 95 to 96 % Cu, 4 to 5 % Mn and small amounts of iron, carbon and leads. It is not suitable for high stress and temperature it may be cold drawn and cold rolled.

Phosphor Bronze

This is copper-tin alloy with a small amount of phosphorous. Its composition is 86 % Cu, 14 % tin and 1 % phosphorous in hard bronze.

Monel Metal

The composition of Monel metal is 67 % Ni and 28 % Cu with a small amount of iron, carbon, manganese. It is resistant to corrosion and is suitable for high temperature. It is used for marine work.

Mild steel

In the past, it was used in many turbines but now it is rarely used. It Corrodes very soon with wet steam but is very inexpensive.

Nickel steel

This alloy is more resistant to corrosion than is mild steel. It forged or machined but not welded. Generally, steel with 3 to 5 % Ni is used. It is used in many turbines.

Stainless steel

It is an alloy of iron, chromium and carbon containing 12 to 14% chromium and normal percentage of carbon. It is very resistant to corrosion and erosion. It is also very hard. It may be rolled, easily, mac hind and welded.

CHAPTER 4:

METHODOLOGY

4 METHODOLOGY

The specification may contain all the necessary details information on absolute flow angle, absolute velocity, diameter of blade design, speed of turbine, whirl flow angle, relative velocity, relative flow angle, Temperature of gases at inlet, velocity of gases at exits, type of material. Turbo machine rotor blades are subjected to different type of loading such as gas forces inertia loads and forces, the selection criteria are that the rotor blade must withstand various stresses due to these forces. The present thesis deals with the stress analysis of the blade made of three materials (Titanium Alloy, Inconel 718 and Structural Steel) which is subjected to centrifugal force loading. A proper design of blade depended on the selection of material. The material of blade plays an important role in the blade design. There are many factories that influence on the criteria of selection blade material like, operation condition, resistance to thermal and mechanical stresses, cost, corrosion, the ability to produce blade free from flaws.

4.1 INTRODUCTION TO CAD

Computer-aided design (CAD) is the use of computer systems (or workstations) to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. The term CADD (for Computer Aided Design and Drafting) is also used.

Its use in designing electronic systems is known as electronic design automation, or EDA. In mechanical design it is known as mechanical design automation (MDA) or computer-aided drafting (CAD), which includes the process of creating a technical drawing with the use of computer software.

CAD software for mechanical design uses either vector-based graphics to depict the objects of traditional drafting, or may also produce raster graphics showing the overall appearance of designed objects. However, it involves more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions.

CAD may be used to design curves and figures in two-dimensional (2D) space; or curves, surfaces, and solids in three-dimensional (3D) space.

CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding, and aerospace industries, industrial and architectural design, prosthetics, and many more.

CAD is also widely used to produce computer animation for special effects in movies, advertising and technical manuals, often called DCC digital content creation. The modern ubiquity and power of computers means that even perfume bottles and shampoo dispensers are designed using techniques unheard of by engineers of the 1960s. Because of its enormous economic importance, CAD has been a major driving force for research in computational geometry, computer graphics (both hardware and software), and discrete differential geometry

The design of geometric models for object shapes, in particular, is occasionally called computer-aided geometric design (CAGD).

TYPES OF CAD

There are several different types of CAD, each requiring the operator to think differently about how to use them and design their virtual components in a different manner for each.

There are many producers of the lower-end 2D systems, including a number of free and open source programs. These provide an approach to the drawing process without all the fuss over scale and placement on the drawing sheet that accompanied hand drafting since these can be adjusted as required during the creation of the final draft.

3D wireframe is basically an extension of 2D drafting (not often used today). Each line has to be manually inserted into the drawing. The final product has no mass properties associated with it and cannot have features directly added to it, such as holes. The operator approaches these in a similar fashion to the 2D systems, although many 3D systems allow using the wireframe model to make the final engineering drawing views.

3D "dumb" solids are created in a way analogous to manipulations of real-world objects (not often used today). Basic three-dimensional geometric forms (prisms, cylinders, spheres, and so on) have solid volumes added or subtracted from them as if assembling or cutting real-world objects. Two-dimensional projected views can easily be generated from the models. Basic 3D solids don't usually include tools to easily allow motion of components, set limits to their motion, or identify interference between components.

There are two types of 3D Solid Modeling

Parametric modeling allows the operator to use what is referred to as "design intent". The objects and features created are modifiable. Any future modifications can be made by changing how the original part was created. If a feature was intended to be located from the center of the part, the operator should locate it from the center of the model. The feature could be located using any geometric object already available in the part, but this random placement would defeat the design intent. If the operator designs the part as it functions the parametric modeler is able to make changes to the part while maintaining geometric and functional relationships.

Direct or Explicit modeling provide the ability to edit geometry without a history tree. With direct modeling, once a sketch is used to create geometry the sketch is incorporated into the new geometry and the designer just modifies the geometry without needing the original sketch. As with parametric modeling, direct modeling has the ability to include relationships between selected geometry (e.g., tangency, concentricity).

Top end systems offer the capabilities to incorporate more organic, aesthetics and ergonomic features into designs. Freeform surface modeling is often combined with solids to allow the designer to create products that fit the human form and visual requirements as well as they interface with the machine.

4.2 JUSTIFICATION OF USING MESHING USING ANSYS

It is a solution phase of the analysis, the computer takes over and solves the simultaneous equations that the finite element method generates. The results of the solutions are

a) Nodal degree of freedom values form the primary solution and

b) Derived values, which form the element solution.

The element solution is usually calculated at the element-integration points. several methods of solving the simultaneous solutions are available in the ANSYS Program, frontal solution, sparse direction solution, Jacobi Conjugate, Gradient solution, Precondition Conjugate Solution and an automatic iteration solver option. The frontal solver is the default.

M0DEL GENERATION

The ultimate purpose of a finite element analysis is to re-create mathematically the behavior of an actual engineering system. In other Words, the analysis must be an accurate mathematical model of a physical prototype. In the broader sense, this model comprises all the nodes, elements, material properties, real constants, boundary conditions and other features that are used to represent the physical system. In ANSYS terminology, the term model generation usually takes on the narrower meaning of generating the nodes and

elements that represent the spatial volume and connectivity of the actual system.

The ANSYS program offers you the following approaches to model generation.

- **Creating a solid model within ANSYS.**
- **Using direct generation.**
- **Importing a model created in a computer-aided design (CAD) system.**

4.3 TYPICAL STEPS INVOLVED IN MODEL GENERATION WITHIN ANSYS

A common modelling session follows this general outline:

Begin by planning your approach - Determine your objectives, decide what basic form your model will take, choose appropriate element types and consider how you will establish an appropriate mesh density.

Enter the preprocessor to initiate your model building session.

Establish a working plane.

Generate basic geometric features using geometric primitives and Boolean Operators.

Activate the appropriate coordinate system.

Generate other solid model features from the Bottom Up.

That is creating key points, and then defines lines, areas, and volumes as needed.

Create a table of element attributes (element types, real constants, material properties and element coordinate system).

Set element attribute pointers.

Set meshing controls to establish your desired mesh density.

Create nodes and elements by meshing your solid model.

Save your model as the job_name.db

CHAPTER 5:

EVALUATION OF GAS FORCES ON TURBINE BLADE

CALCULATIONS

Gas Forces acting on the blade of the rotor in general have two components namely Tangential force(F_t) and Axial force(F_a). These forces result from the gas momentum changes and from pressure differences across the blades. These gas forces are evaluated by constructing velocity triangles at inlet and outlet of same profile is taken throughout the length of the blade. If the gas forces are assumed to be distributed evenly then the resultant acts through the Centroid of the area.

Table 5.1

Composition of TITANIUM ALLOYS	
Composition	**Weight Percentage**
Aluminum(Al)	**8%**
Molybdenum(Mo)	**1%**
Titanium(Ti)	**90%**
Vanadium(V)	**1%**

Table 5.2

Properties of TITANIUM ALLOYS	
Density	**4620 Kg/m^3**
Yield Strength	**930 MPa**
Ultimate Strength	**1000 MPa**
Poisson Ratio	**0.36**
Young's Modulus	**96 GPa**
Melting Point	**15400C**
Thermal Expansion	**0.000010/mC**
Thermal Conductivity	**600 w/m.K**
Specific Heat	**502 J/Kg.K**

COMPOSITION OF INCONEL 718	
Composition	Weight Percentage
Chromium(Cr)	17-21%
Iron(Fe)	17%
Molybdenum(Mo)	2.8-3.3%
Nickel(NI)	50-55%

COMPOSITION OF STRUCTURAL STEEL	
Composition	Weight Percentage
Iron(Fe)	98%
Carbon©	0.25%
Sulphur(S)	0.5%
Silicon(Si)	0.5

Table 5.3 and Table 5.4

Properties of INCONEL 718	
Density	8190 Kg/m^3
Yield Strength	870 MPa
Ultimate Strength	1060 MPa
Poisson Ratio	0.284
Young's Modulus	205GPa
Melting Point	1400C
Thermal Expansion	0.000014/mC
Thermal Conductivity	400 w/mK
Specific Heat	352 J/KgK

Properties of STRUCTURAL STEEL	
Density	7850 Kg/m^3
Yield Strength	250 MPa
Ultimate Strength	460 MPa
Poisson Ratio	0.3
Young's Modulus	200GPa
Melting Point	900C
Thermal Expansion	0.000012/mC
Thermal Conductivity	200 w/mK
Specific Heat	105 J/KgK

Table 5.5 and Table 5.6

5.1 EVALUATION OF GAS FORCES ON THE FIRST STAGE ROTOR BLADE

The material is titanium alloy and structural and thermal properties at gas room and room temperature were taken from the design data book that were available in the library of BHEL(R&D), HYDERABAD.

At the Inlet of the First Stage Rotor Blades,

Absolute Flow Angle α_2 = 23.85^0

Absolute Velocity V_2 = 462,21 m/s.

Diameter of Blade mid-span D_m = 1.3035m.

Design Speed of Turbine N = 3426 RPM.

Number of Blade passages in First stage rotor n = 120

Blade angle at inlet θ_2 = 135.017^0.

The velocity triangles at inlet of first stage rotor blades were constructed as shown.

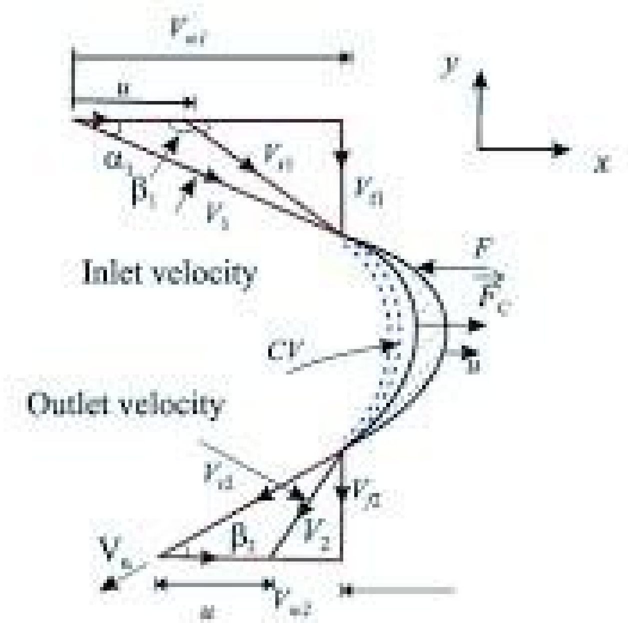


Fig 5.1 Velocity Triangles

Peripheral speed of rotor blade at its mid span U = (Л D N)/60.

= Л * 1.3035 * 3426 / 60

= 233.828 m/s.

From the velocity triangles in figure we get,

Whirl Velocity $V_{w2} = V_2 * \cos \alpha_2$.

= 462.21 * cos(23.85)

= 422.74 m/s.

Flow Velocity $V_{f2} = V_2 * \sin\alpha_2$.

= 462.21 * sin(23.85)

= 186.89 m/s.

Relative Velocity $V_{r2} = \sqrt{(\qquad) \quad (\quad)}$

= 265.735 m/s.

At the Exit of First Stage Rotor Blades,

Flow Velocity, V_{f3} = 180.42 m/s.

Relative Flow Angle, $\phi_3 = 37.88^o$.

From the velocity triangles, we get

$\sin \phi_3 = V_{f3} / V_{r3}$.

Relative Velocity V_{r3}

$= V_{f3} / \sin \phi_3$

= 293.83 m/s.

Whirl Velocity $V_{w3} = U – (V_{r3} * \cos \phi_3)$

= 2.805 m/s.

Finding Tangential and Axial Forces on each Rotor.

Tangential Force

$F_T = M(V_{w2} – (+V_{w3})$

Axial Force

$$F_A = M(V_{f2} - (+V_{f3})$$

Where M, represents mass flow rate of gases through the turbine in Kg/sec.

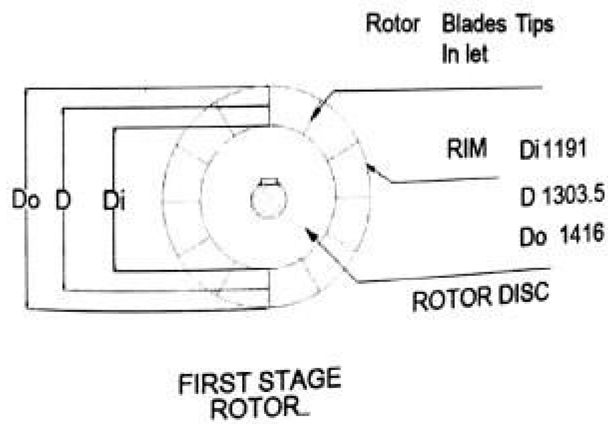


Fig 5.2 First Stage Rotor

The mass flow rate of gases through the turbine in Kg/s, $M = \rho_2 * \pi * (D_0^2 - D_1^2)/4 * V_{f2}$

$$M = 0.823 * \pi * (1416^2 - 1191^2)/4 * 186.89.$$

$$M = 70.894 \text{ Kg/sec.}$$

Total Tangential Force on First Stage Rotor:

$$F_T = 70.894 * [(422074 - (+2.801)]$$

$$F_T = 29771.25N$$

Total Number of Blades $n = 120$,

Tangential Force on each blade

$$F_T = 458.88 \text{ N.}$$

Total Axial Force on First Stage Rotor:

$$F_A = 70.894 * [(186.89 - (+180.42)]$$

$$F_A = 458.34N$$

Total Number of Blades $n = 120$,

Axial Force on each blade $F_T = 3.819N$

Power Developed(P) in the First Stage Rotor Blade.

From Euler's Energy Equation

$$P = m\{V_{f2}U - (+V_{f2}U)\}$$

$$= 70.42*234.606*\{422.74 - (+2.805)\}$$

$$= 6986986.672 \text{ Watts}$$

$$= 6.9869 \text{ MW.}$$

$$V = 79311.37 \text{ mm}^3.$$

$$V = 0.00007931137 \text{ m}^3.$$

Centrifugal Force $F_c = M * (2 * \pi * N/60) * X$.

Where M is Mass of Blade

X is Distance from Axis of Revolution to Centroid.

N is RPM.

	Titanium Alloy	Inconel 718	Structural Steel
Density(Kg/m^3)	4620	8190	7850
Volume(m^3)	0.00007931137	0.00007931137	0.00007931137
Mass(Kgs)	0.366	0.64956	0.6225
Centrifugal Force(kN)	29.921	53.103	50.891

Table 5.7 Mass and Centrifugal Force for Three Materials

5.2 CONVECTION HEAT TRANSFER COEFFECIENTS OVER THE BLADE SURFACE.

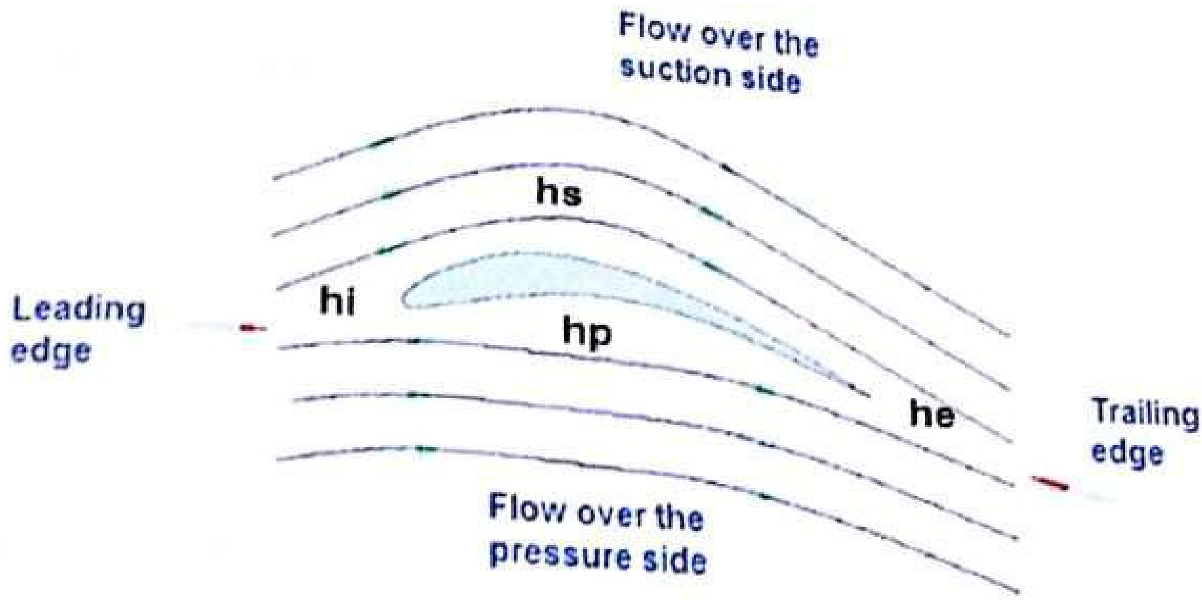


Fig. 5.3 Flow over Aero Foil Profile

EVALUATION OF CONVECTIVE HEAT TRANSFER COEFFICIENTS (h_s) ON SUCTION SIDE OF FIRST STAGE ROTOR BLADES

For first stage rotor blades, as shown

Temperature of gases at inlet, T_i = 839.22°C

Temperature of gases at exist, T_e = 732.88°C

Mean fluid temperature

$$T_{mf} = (T_i + T_e)/2$$

$$T_{mf} = 785.49°C\ .$$

The flowing properties of air at T_{mf} were noted in heat and mass transfer data book (seventh edition) Kinematic viscosity (v) = 1 1 9 * 10^{-6} m2/sec
Prandtl Number (Pr) =0.0712

Thermal Conductivity (K) =0.0712 W/m.K

On suction side the flow is approximated to the flow across a cylinder whose diameter (D_{ch}) is equal to the chord length of the aero foil. From the design data book of BHEL, Hyderabad.

D_{ch} = 50.4 mm

Average relative velocity

$$V_r = (V_{r2} + V_{r3})/2$$

$$V_r = 279.97 m/s.$$

Reynolds number Re = $\frac{*}{\quad}$

$$Re = 118499.29$$

The Nusselt number (Nu) is calculated using the empirical relation for flow across cylinders.

The generalized equation is

$$NuD = C\ ReD^{m} Pr^{0.333}$$

The values of C and M are selected from the heat and mass transfer Data book Seventh Edition (C.P.Kothandaraman) table below

ReD	C	M
0.4 – 4.0	0.989	0.330
4.0 – 4.0	0.911	0.385
40.0 – 4000	40.0 – 4000	0.4666
4000 – 40000	0.193	0.618
40000 – 4000000	0.0266	0.805

Table 5.8 Values of C and M for Varying Reynolds Number

Where

(M) Coefficient = 0.805

(C) Constant = 0.0266

Now, NuD = 0.0266* 1 1 8499.29 * $(50.4*10^{-3})^{0.805}$ * $(0.712)^{0.333}$ =254.05

NuD = ――――――

H_s = 360.4 W/m^2K

EVALUATION OF CONVECTIVE HEAT TRANSFER COEFFICIENTS (h_p) ON THE PRESSURE SIDE OF FIRST STAGE ROTOR BLADES

For first stage rotor blades

T_{mf} = $(T_i + T_e)/2$

T_{mf} = 785.49 °C

Kinematic Viscosity (v), Prandtl Number (Pr) and Thermal Conductivity (k) of air at Lf are noted down from air tables on pressure side; the flow is approximated to the flow over a flat plate whose length (L) along the flow direction is equal to chord length (1) of the aero foil blade.

Kinematic viscosity (v) =119 * 10^{-6} m^2/sec

Prandtl Number (Pr) = 0.712

Thermal Conductivity (K) =0.0702 W/m.K

Reynolds Number (Re) = 118400.29

Since Re < 5 * 10^5, Flow is Laminar on the pressure side, Following is the empirical relation for Laminar flow over a flat plate,

$Nu = 0.644 * Pr^{1/3} * Re^{1/2}$.

Nu = 204.1

NuD = $(h_p + D_{ch})/k$

H_p = 284.28 W/m^2K.

EVALUATION OF CONVECTIVE HEAT TRANSFER COEFFICIENTS (h_r) AT INLET AND EXIT OF FIRST STAGE ROTOR BLADES

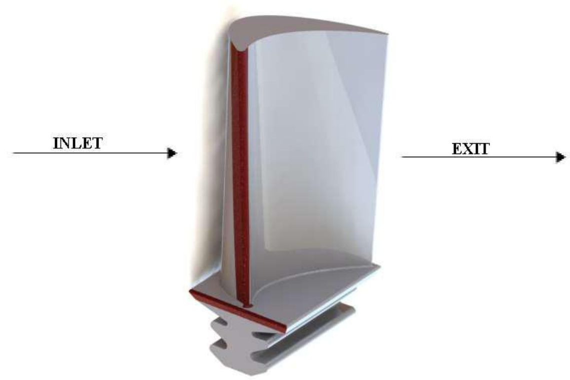


Fig. 5.4 Convective Heat Transfer Co-efficient at Inlet

Peripheral speed of rotor blade at its mid span U = Л D N/60

= Л * 1 .3035* 3426/60

=233.828 m/s

The flow can be approximated to the flow over a flat plate whose length is equal to the length (w) of the rectangular face.

Gas temperature at inlet, T_i = 839.22°C

v, Pr and K of air are noted at temperature T_i from the air table.

Kinematic viscosity (v) =148.58 $*10^{-6}$ m^2/sec

Prandtl Number (Pr) = 0. 713.

Thermal Conductivity (K) =0.0724 W/m.K

Reynolds number Re =21245

For laminar flow over a flat plate using equation

$$Nu = 0.644 * Pr^{1/3} * Re^{1/2}$$

$$Nu = 0.664 * 0.713^{1/3} * 21245.1^{1/2}$$

$$= 86.46$$

For Flat Plate, NuD = (h_i + w)/k

$$H_i = 231.9\ W/m^2.K$$

AT EXIT:

Fig. 5.5 Convective Heat Transfer Co-efficient at Exit

The gases sweep the faces with a velocity,

Peripheral speed of rotor blade at its mid span U = Л DN/60

= Л * 1.3035 * 3426/60

=233.828 m/s.

The flow can be approximated to the flow over a flat plate whose length is equal to the length (w) of the rectangular face.

Gas temperature at exist, T_e = 732°C

v, Pr and K of air are noted at temperature T_e from the air table.

Kinematic viscosity (v) =124.86 $*10^{-6}$ m^2/sec.

Reynolds number Re =25233.51

For laminar flow over a flat plate using equation

$$Nu = 0.644 * Pr^{1/3} * Re^{1/2}$$

$$Nu = 0.664 * 0.713^{1/3} * 25233.51^{1/2}$$

$$= 93$$

For Flat Plate, NuD = $(h_e + w)/k$

$$H_e = 228.4 \ W/m^2.K.$$

MATERIAL	F_A	F_C	F_T	h_e	h_i	h_p	h_s
TITANIUM ALLOY	458.88N	29921.44N	3.819N	228.4 $W/m^2.K.$	231.9 $W/m^2.K.$	284.28 $W/m^2.K.$	360.4 $W/m^2.K.$
INCONEL 718		53103.19N					
STRUCTURAL STEEL		50890.97N					

<u>**Table 5.9 All Forces and All Convective Heat Transfer Co-efficient on Blade (for 3 materials)**</u>

CHAPTER 6:

MODELING AND ANALYSIS

6.1 MODELING USING SOLIDWORKS .

First, the Aero foil profile was generated on the XY Plane with the help of co-ordinates from the NACA 65 Series aero foil profiles. Then a number of splines were fitted through the key points. A rectangle of dimensions 63.15 * 43.32 mm was generated as shown.

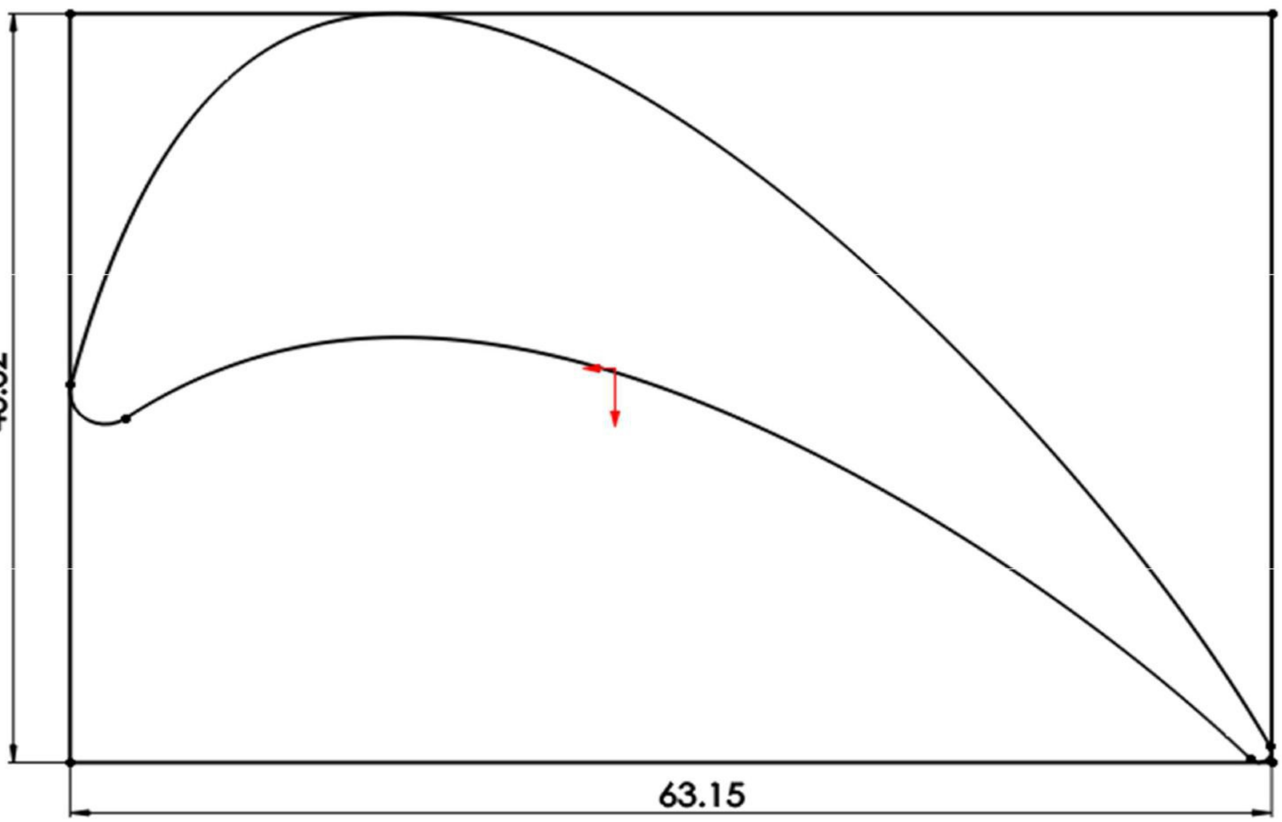
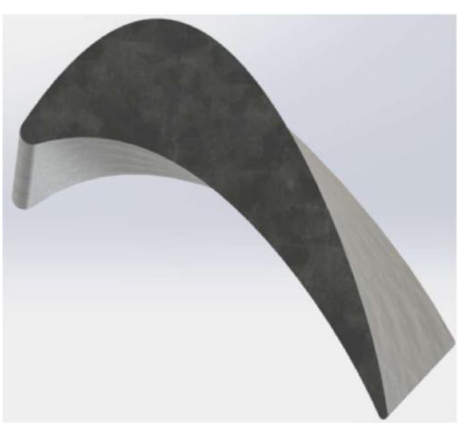


Fig. 6.1 Gas Turbine Blade Aero Foil Profile using NACA 65 series.

The profile was lofted 117mm, as per BHEL(R&D) Design Data book dimensions, and twisted with an angle of 48º.



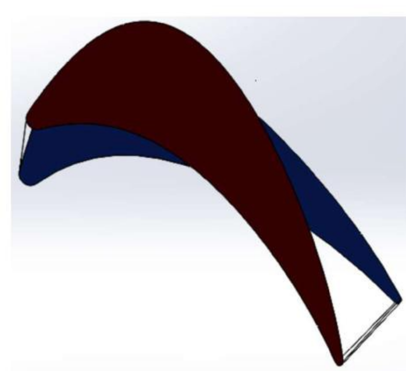
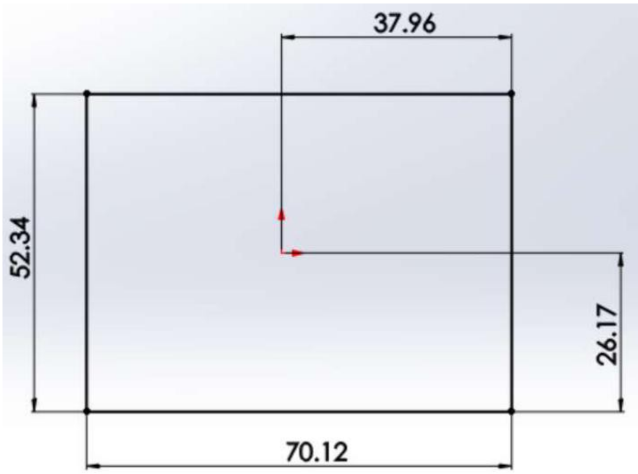


Fig 6.2 Different Views of Lofted Profile.

Create a Base rectangle of dimensions 70.12 * 52.34 mm, extredu it for 5mm and with a draft of 30º.



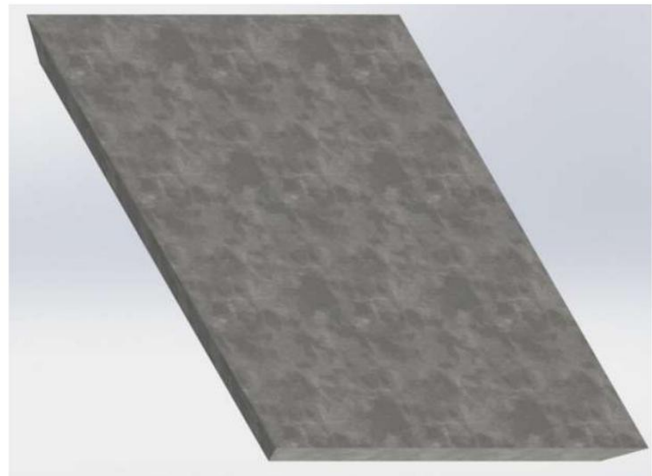
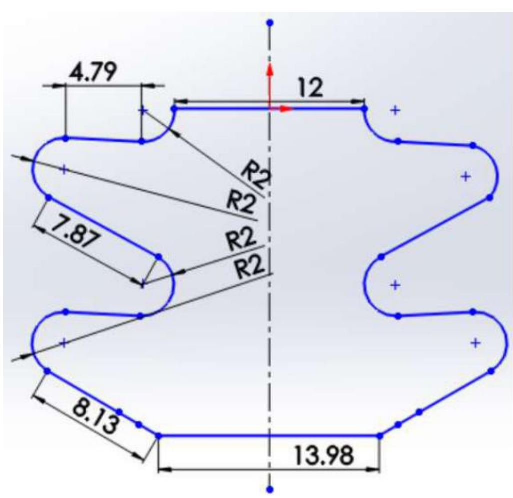


Fig. 6.3 Base of Lofted Blade

Create the base insert of the blade, which will be used in inserting the blade into the rotor disk.



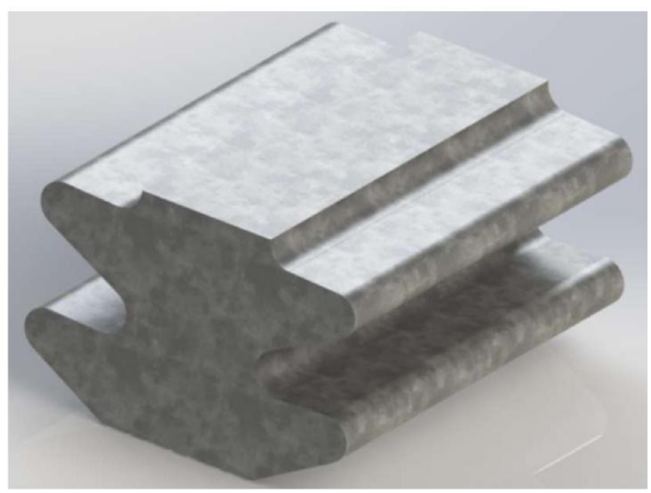


Fig. 6.4 Extrude of Base Insert.

Give Fillets of 1mm to all the sharp edges, to avoid Stress Concentration.

Save the file as "Blade.step"

IMAGES OF THE GAS TURBINE BLADE MODEL.

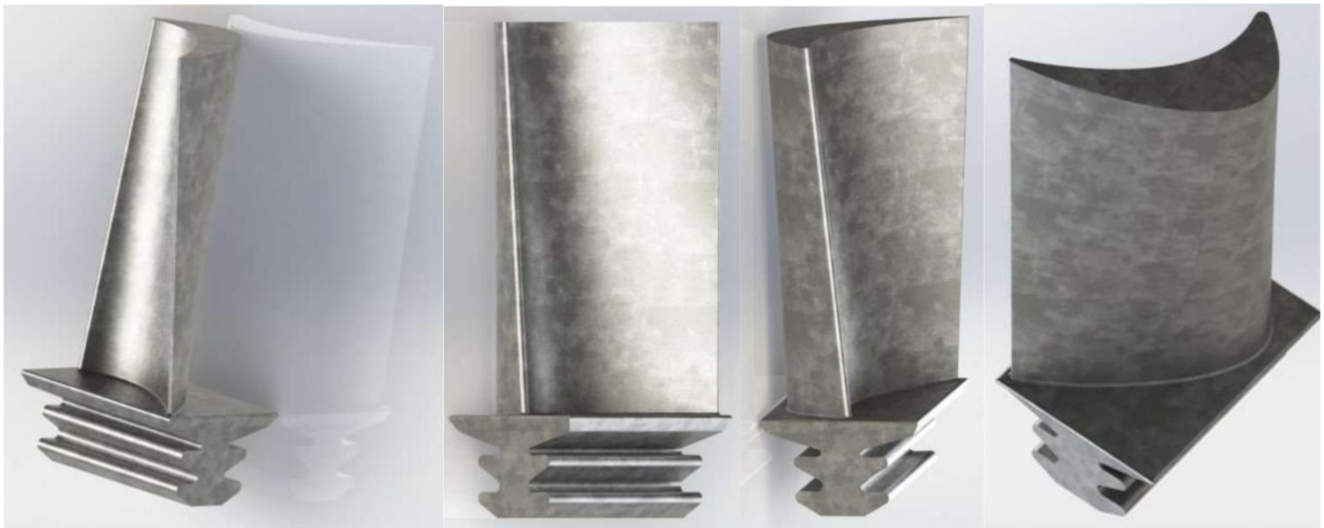


Fig 6.5 Different Views of Blade (Inconel 738).

Fig 6.6 Different Views of Blade (Structural Steel).

Fig 6.6 Different Views of Blade (Titanium Alloy).

6.2 ANALYSIS USING ANSYS.

6.2.1 STRUCTURAL ANALYSIS.

Import the Gas Turbine Blade model, into ANSYS Workbench – Static Structutal, Geometry section. In Mechanical, 3D Meshing has been performed with Hexa meshing done as much as possible and quad mesh in the remaining palces. The meshing was done by keeping the element size as 3mm and the Element Order as Quadratic.

After Meshing the number of nodes and elements are:

Statistics	
Nodes	79624
Elements	52295

Fig 6.8 Mesh Stastics

The Mesh Qualitiy Stastistics are:

Mesh Metric	Element Quality
Min	4.6053e-002
Max	0.99991
Average	0.80885
Standard Deviation	0.13588

Mesh Metric	Aspect Ratio
Min	1.1671
Max	43.296
Average	2.108
Standard Deviation	2.0682

Mesh Metric	Skewness
Min	8.5885e-004
Max	0.99992
Average	0.2654
Standard Deviation	0.16735

Fig 6.9 Mesh Quality Statistics

Two structural boundary conditions namely displacement and force were applied on the rotor blade model. The solution part of ANSYS was opened and the bottom part the blade's displacement was constrained.

U_x U_y U_z = 0. and Rot_x Rot_y Rot_z = 0.

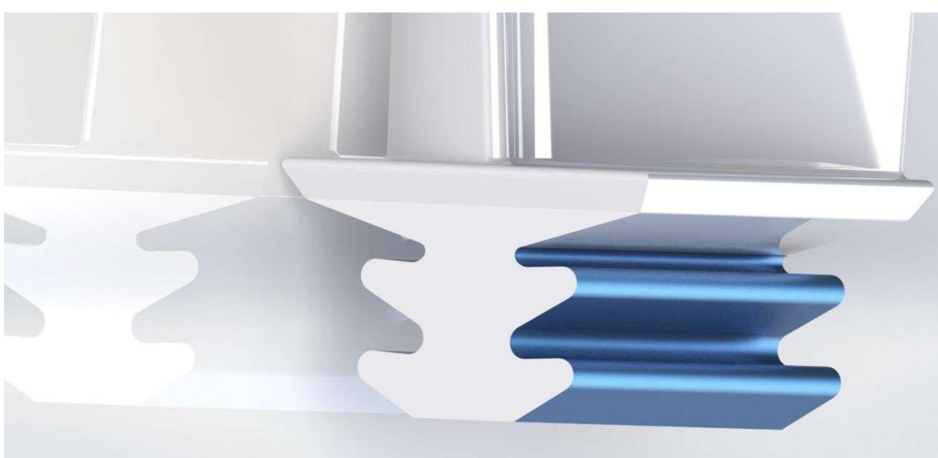


Fig 6.10 Fixed Support for Structural Analysis

The constraint is on both sides of the part visible in the image.

Table 6.1

MATERIAL	F_A	F_C	F_T
TITANIUM ALLOY		29921.44N	
INCONEL 718	458.88N	53103.19N	3.819N
STRUCTURAL STEEL		50890.97N	

In the solution part of ANSYS the blade forces namely Tangential, Axial and Centrifugal forces were applied on the surface of the blade, where gas hits the surface.

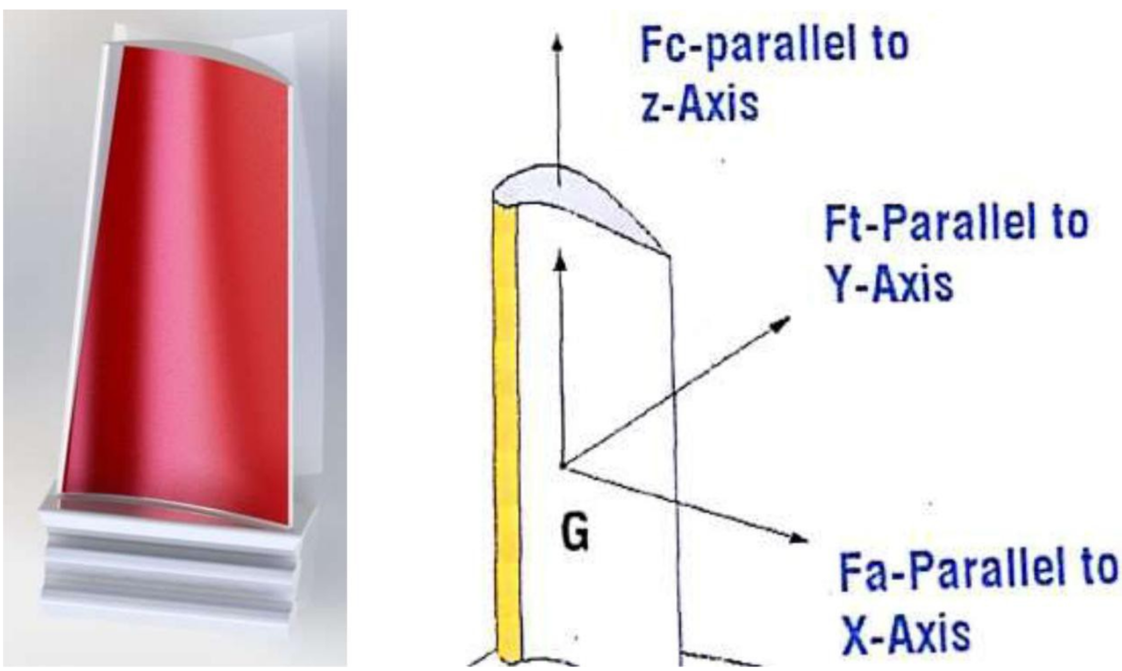


Fig 6.11 Force Acting Surface and Direction of Forces

In the ANSYS Solution, Stress, Deformation and the Safety Factor are found out.

This same steps are performed again by changing the material to Inconel 738 and Structural steel.

The meshing remains the same irrespective of the material used, but the weight changes and also it's properties.

The difference in Stress(Von-misses). Total Deformation and Safety factor for this three materials are shown below.

TITANIUM ALLOY

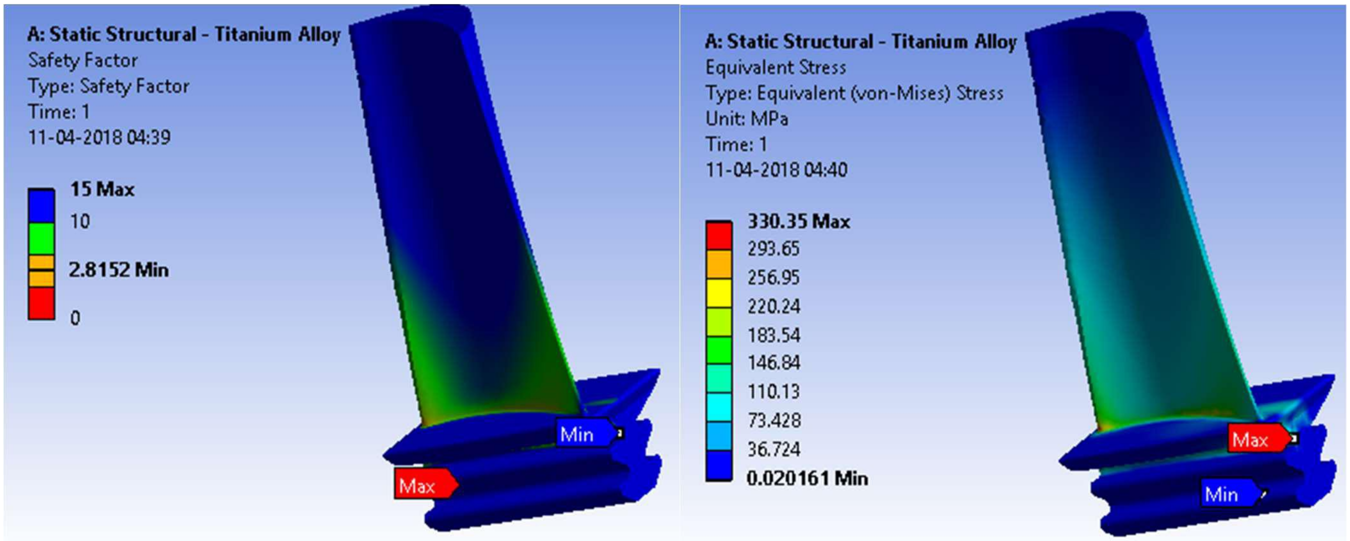


Fig 6.12 & 6.13 – Safety Factor and Equivalent Stress(Von-Misses) for Titanium Alloy

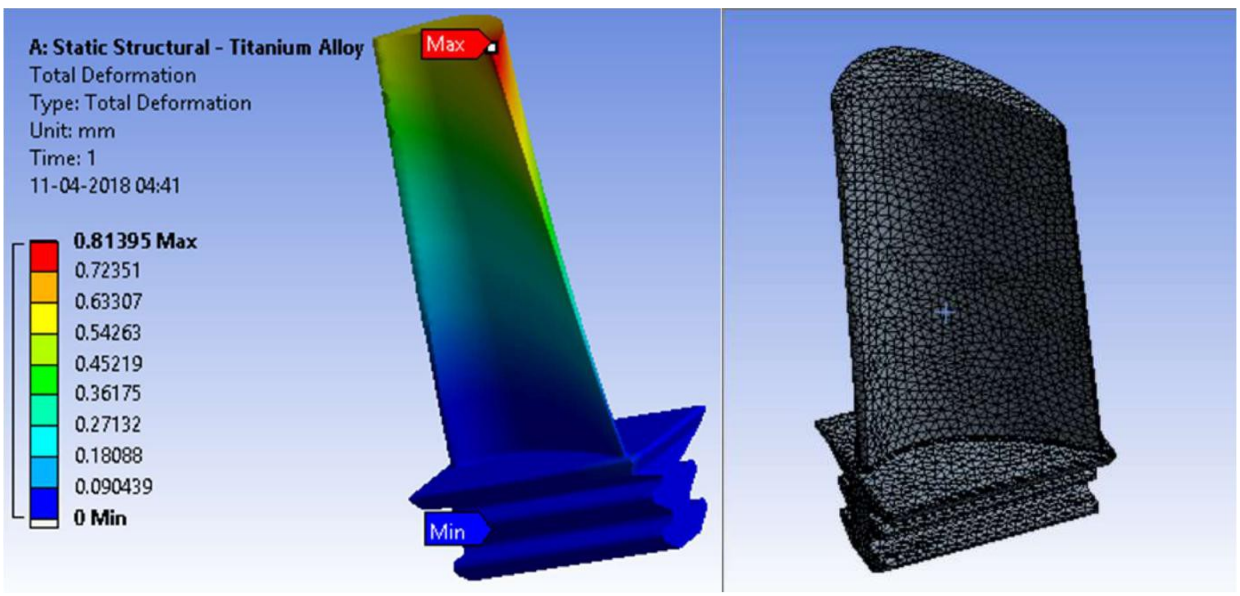


Fig 6.14 & 6.15 – Total Deformation and Mesh for Titanium Alloy

Maximum Deformation = 0.81395 mm(at the tip of the blade) Minimum Factor of Safety = 2.8152 Max Stress(Von-Misses) = 330.35 MPa.

INCONEL 738

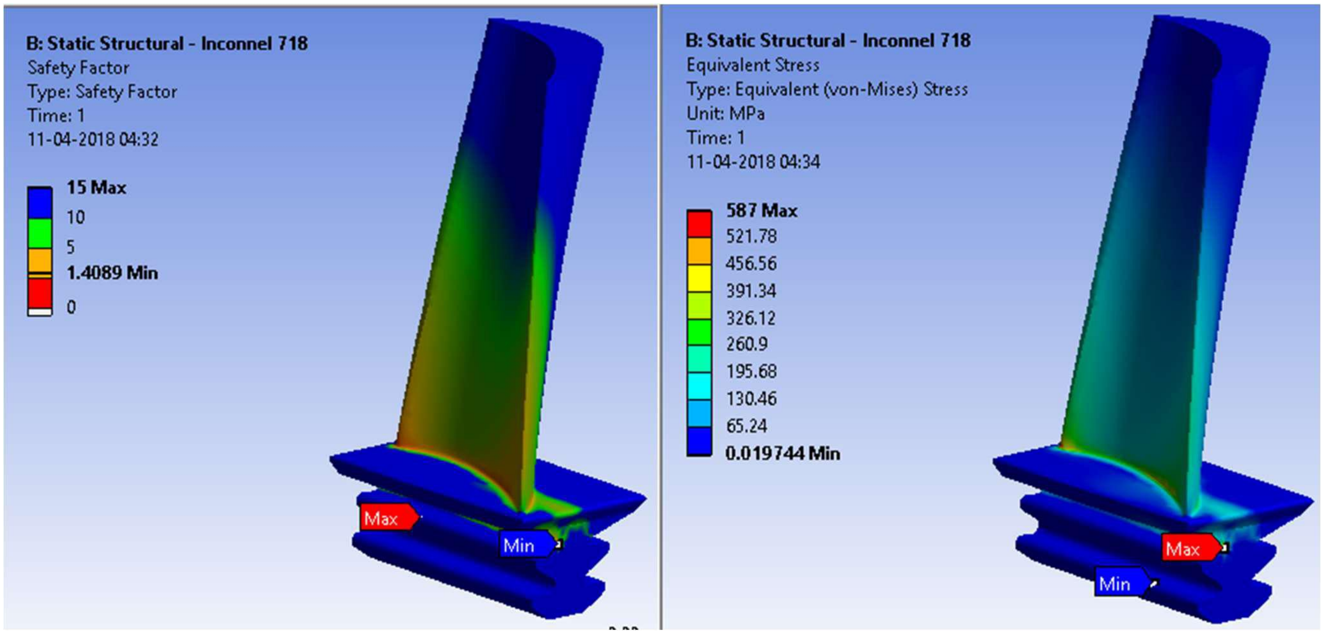


Fig 6.16 & 6.17 – Safety Factor and Equivalent Stress(Von-Misses) for Inconel 738

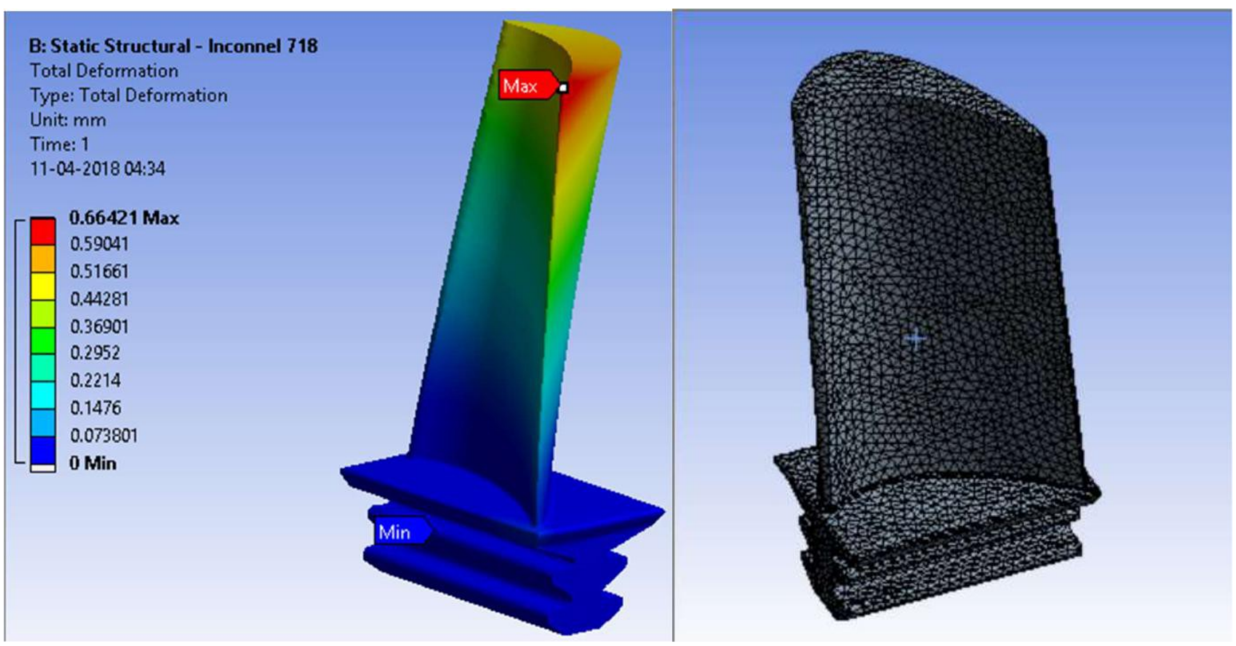


Fig 6.18 & 6.19 – Total Deformation and Mesh for Inconel 738

Maximum Deformation = 0.664215 mm(at the tip of the blade) Minimum Factor of Safety = 1.4089 Max Stress(Von-Misses) = 587 MPa.

STRUCTURAL STEEL

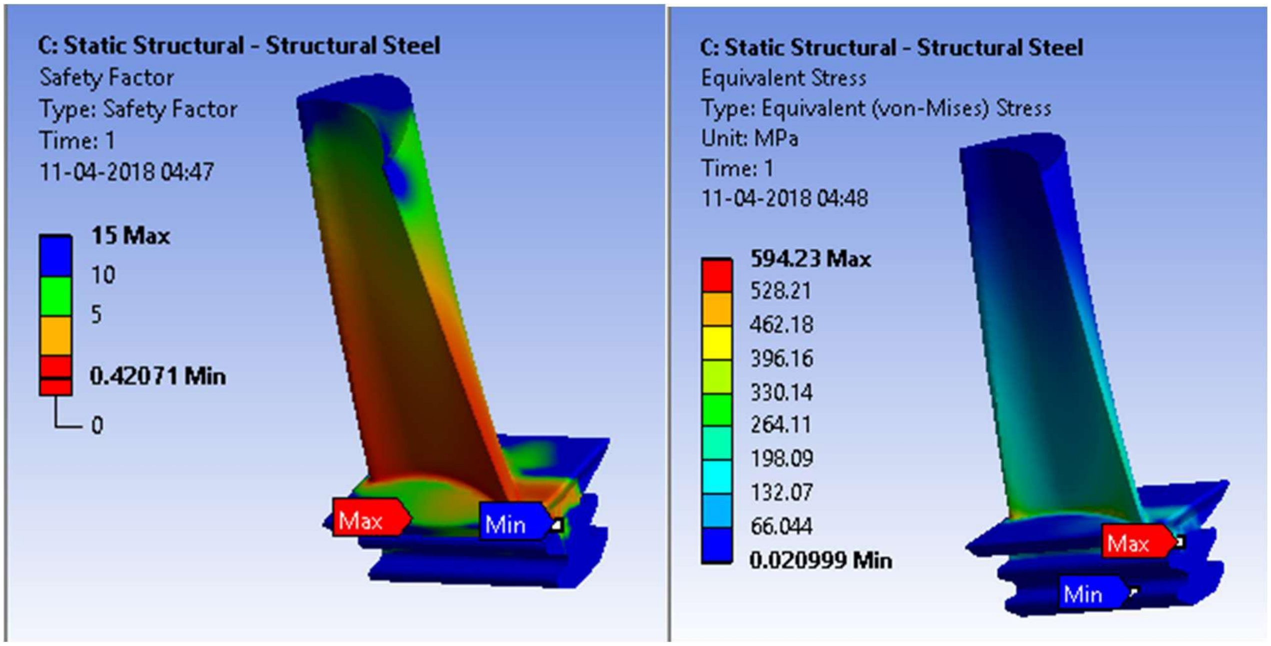


Fig 6.20 & 6.21 – Safety Factor and Equivalent Stress(Von-Misses) for Structural Steel

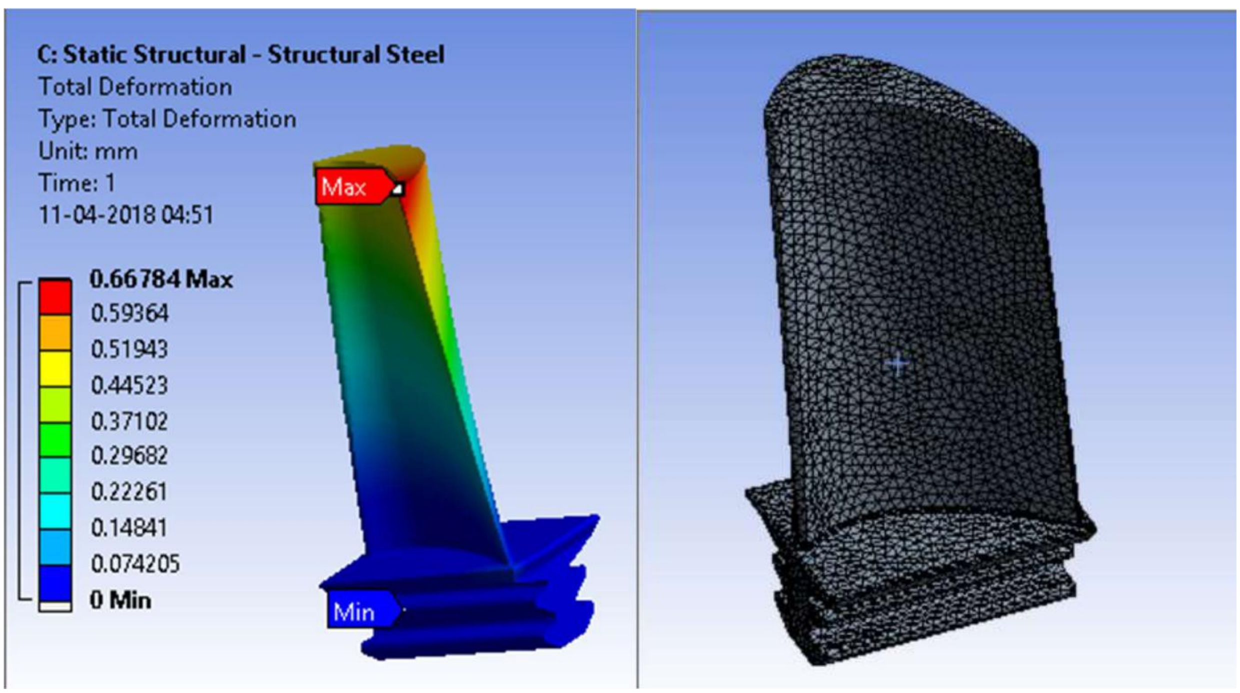
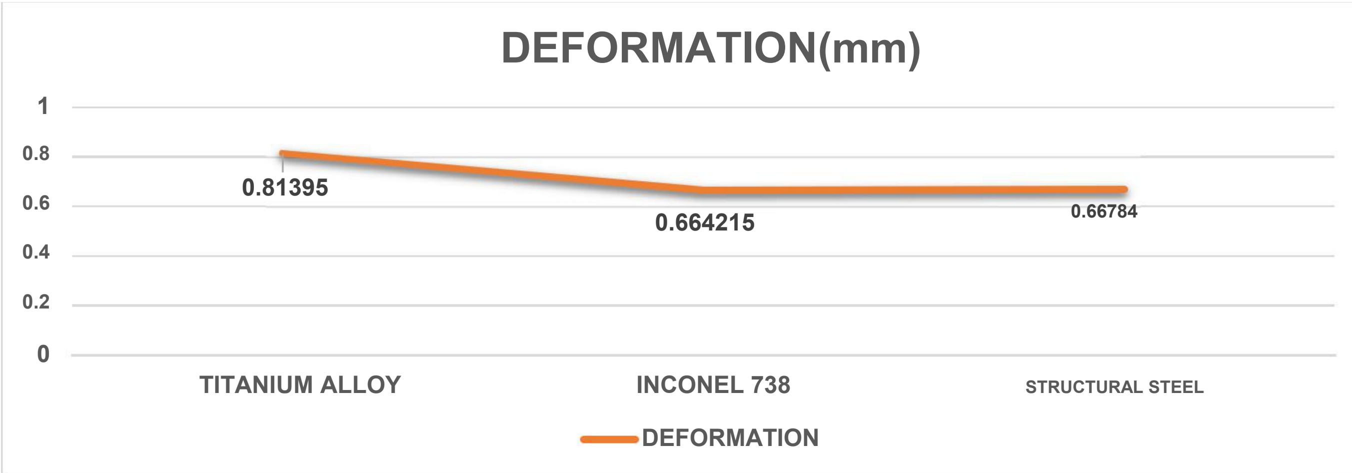


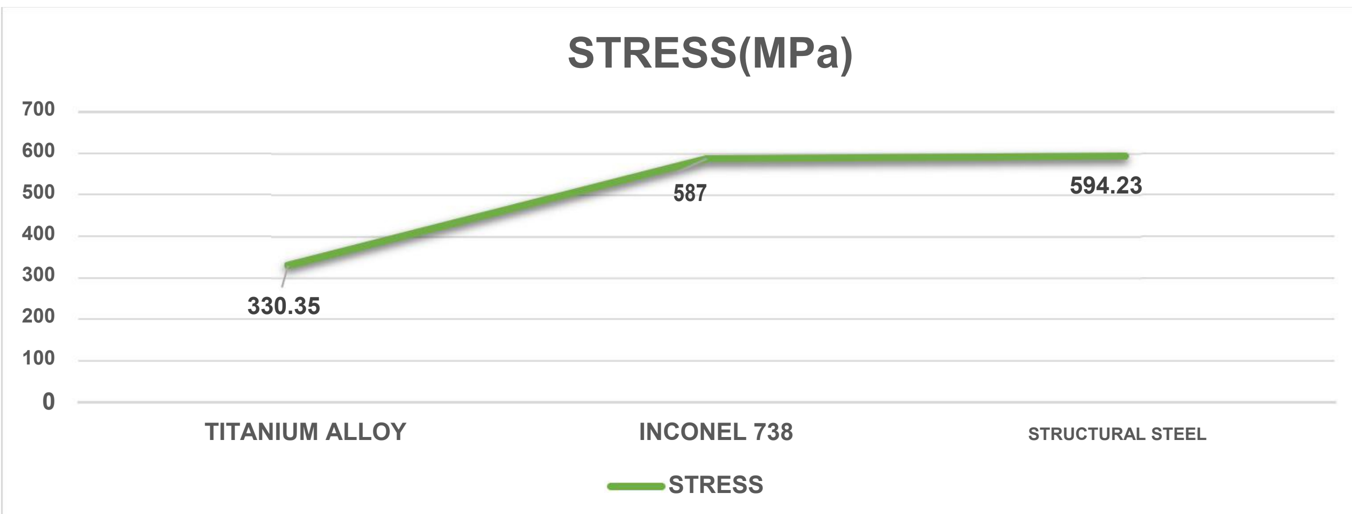
Fig 6.22 & 6.23 – Total Deformation and Mesh for Structural Steel

Maximum Deformation = 0.66784 mm(at the tip of the blade) Minimum Factor of Safety = 0.42071 Max Stress(Von-Misses) = 594.23 MPa.

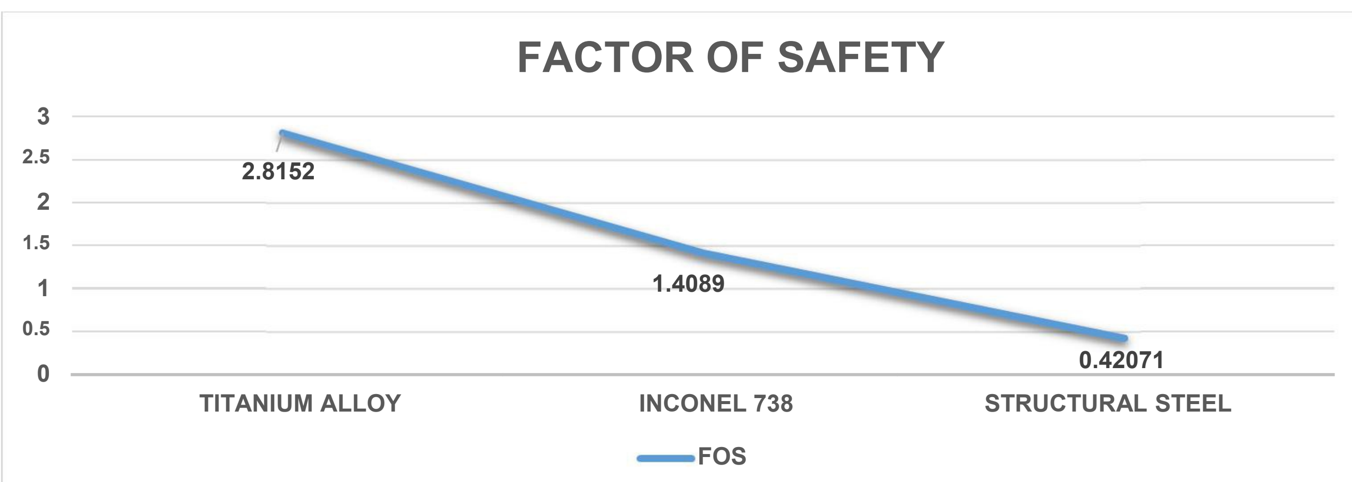
COMPARISION:



Graph 6.1 Deforamtion Comparision



Graph 6.2 Stress Comparision



Graph 6.3 Factor of Safety Comparision

6.2.2 THERMAL ANALYSIS

A new module, named Steady-State Thermal is introduced into the project schematc, and the same geometry is imported into the module.

In Mechanical, 3D Meshing has been performed with Hexa meshing done as much as possible and quad mesh in the remaining palces. The meshing was done by keeping the element size as 3mm and the Element Order as Quadratic.

After Meshing the number of nodes and elements are:

Statistics	
Nodes	79624
Elements	52295

The Mesh Qualitiy Stastistics are:

Mesh Metric	Element Quality
Min	4.6053e-002
Max	0.99991
Average	0.80885
Standard Deviation	0.13588

Mesh Metric	Aspect Ratio
Min	1.1671
Max	43.296
Average	2.108
Standard Deviation	2.0682

Mesh Metric	Skewness
Min	8.5885e-004
Max	0.99992
Average	0.2654
Standard Deviation	0.16735

The two boundary conditions namely Heat Flux and Convection is applied on the rotor blade model The Heat Flux is set to zero, to the complete model.

The area of constraints in static structural analysis which are contact with their adjacent sides, are in symmetry are assumed to be perfectly insulated.

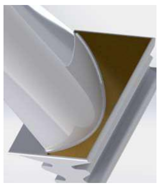
And also the other areas of the bottom part of the turbine being small compared to the blade is very small, so it can be assumed to be insulated as well.In convective boundary condition, the conventicve heat transfer coeffecient and temperatuer of gas in surrondng for diffrenet region can be given as:

$h = h_s + h_p = 322.4 W/m^2K$ and $T = 839.22^oC$.

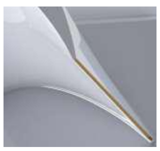
$h_p = 284.28 W/m^2K$ and $T = 786.11^oC$.

h_p = 284.28W/m^{2}K and T = 786.11ºC.

h_s = 360.4 W/m^{2}K and T = 786.11ºC.



h_s = 360.4 W/m^{2}K and T = 786.11ºC.



h_e = 231.19 W/m^{2}K and T = 733ºC.

In the ANSYS Solution, Temperature and Heat Flux are found out.

This same steps are performed again by changing the material to Inconel 738 and Structural steel.

The meshing remains the same irrespective of the material used, but the weight changes and also it's properties.

The difference in Temperature and Heat Flux for this three materials are shown below.

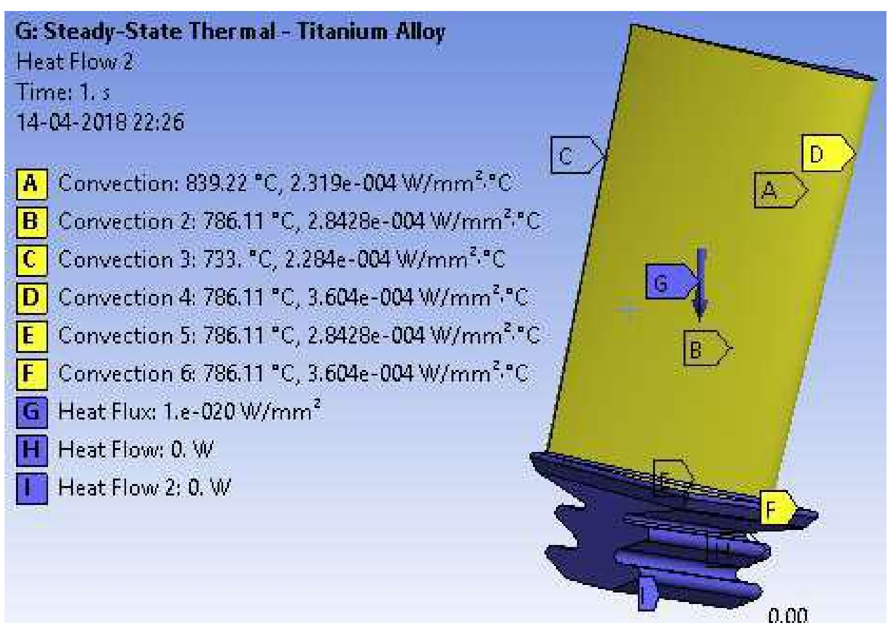
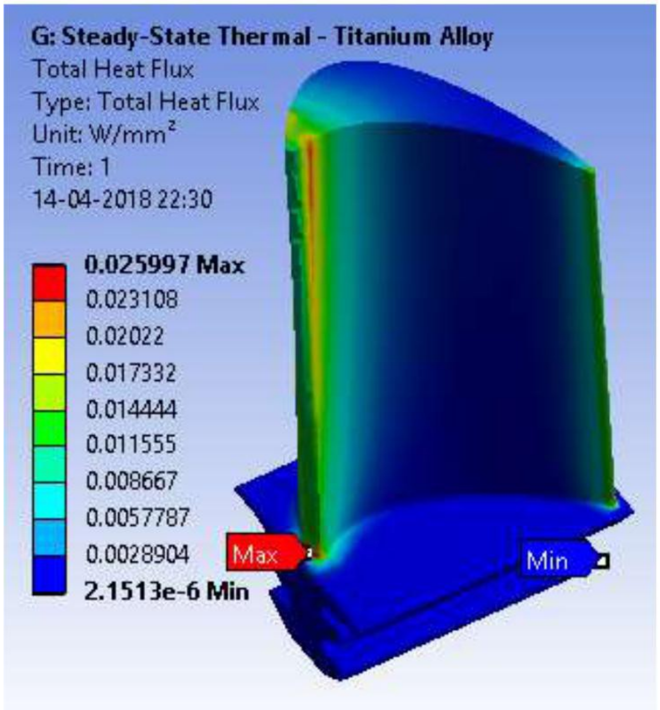


Fig 6.24 Thermal Analysis Boundary Condition

TITANIUM ALLOY



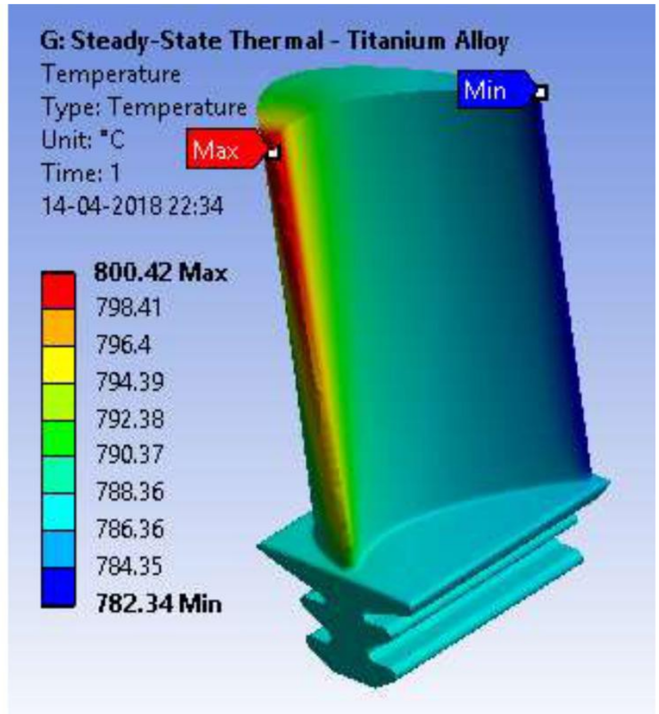
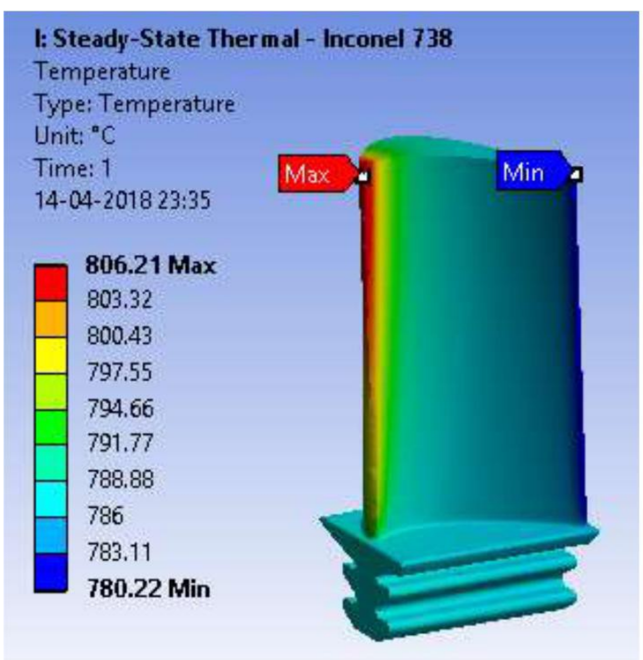


Fig 6.25 and 6.26 – Total Heat Flux and Temperature Distribution for Titanium Alloy.

Maximum Temperature: 800.42ºC

Maximum Total Heat Flux: 0.025997 W/mm^2

INCONEL 738



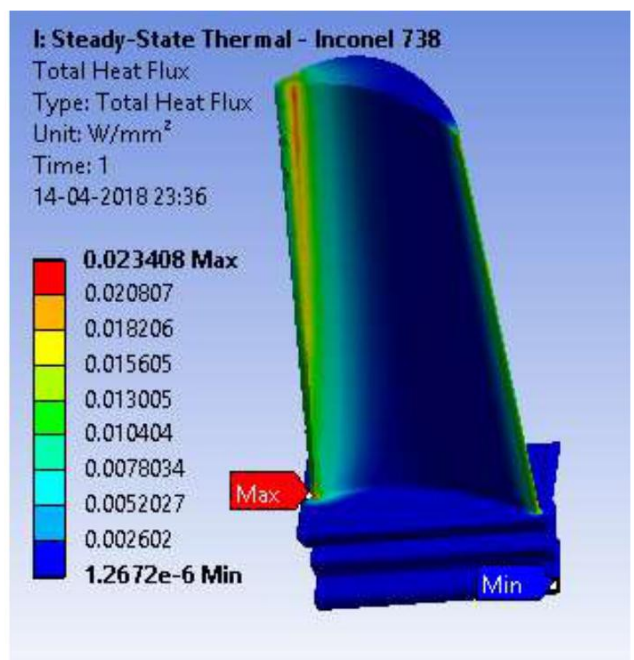
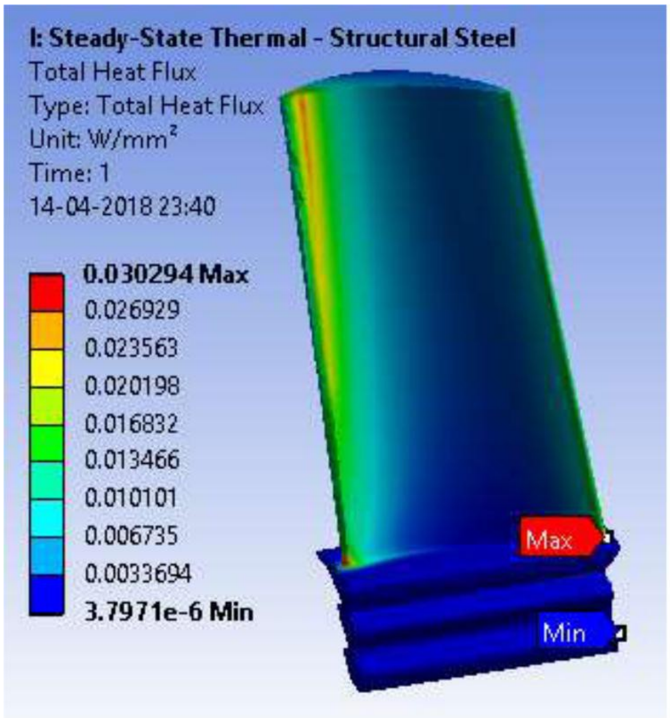


Fig 6.27 and 6.28 – Total Heat Flux and Temperature Distribution for Inconel 738.

Maximum Temperature: 806.21ºC

Maximum Total Heat Flux: 0.023408 W/mm^2

STRUCTUAL STEEL



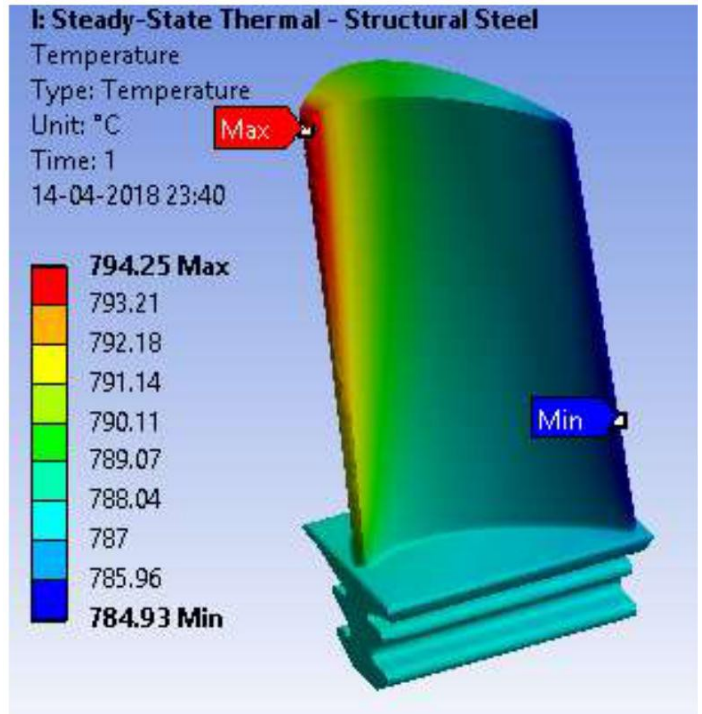
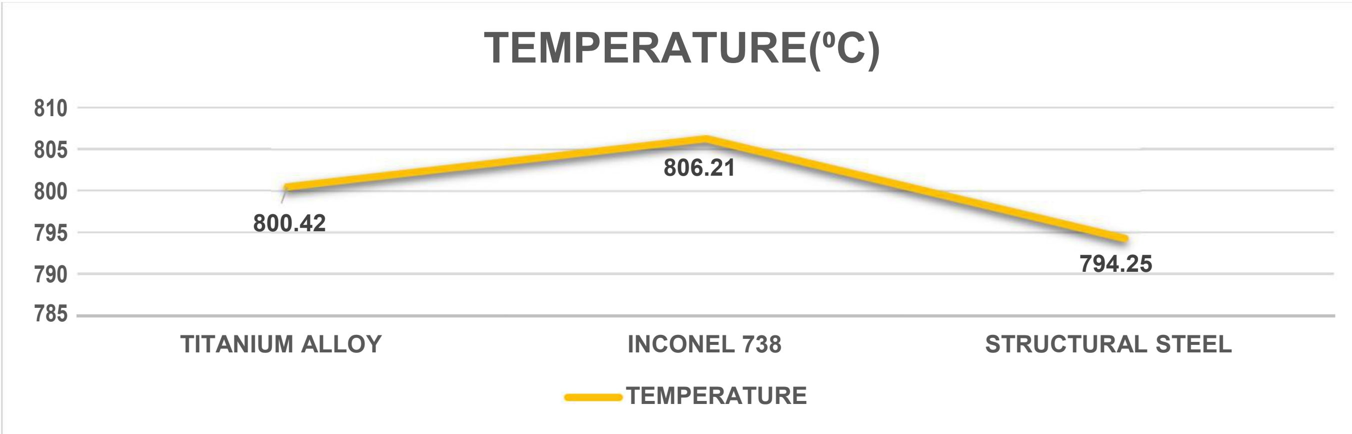


Fig 6.29 and 6.30 – Total Heat Flux and Temperature Distribution for Structural Steel.

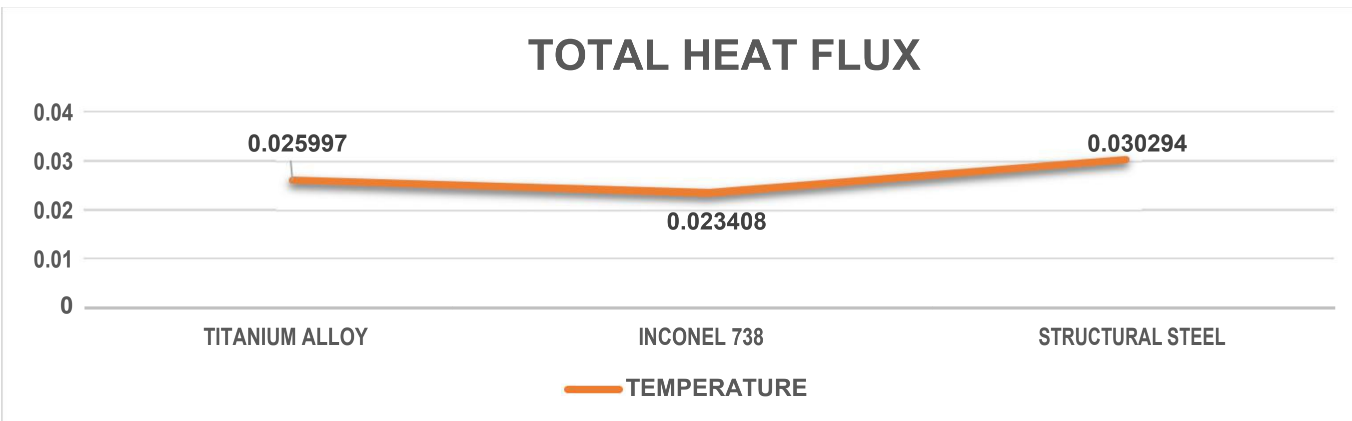
Maximum Temperature: 794.25ºC

Maximum Total Heat Flux: 0.030294 W/mm^2

COMPARISION:



Graph 6.4: Temperature Distribution



Graph 6.5: Total Heat Flux

MATERIAL	DEFORMATION	STRESS	FOS	TEMPERATURE	HEAT FLUX
TITANIUM ALLOY	0.81395	330.35	2.8152	800.42	0.025997
INCONEL 738	0.664215	587	1.4089	806.21	0.023408
STRUCTURAL STEEL	0.66784	594.23	0.42071	794.25	0.030294

Table 6.2: Structural and Thermal Analysis Results

6.2.3 MODAL ANALYSIS

In structural engineering, modal analysis uses the overall mass and stiffness of a structure to find the various periods at which it will naturally resonate. If a structure's natural frequency matches a working frequency the structure may continue to resonate and experience structural damage. Modal analysis is also important in structures where the engineer should attempt to keep the natural frequencies away from the working frequencies.

Import the Gas Turbine Blade model, into ANSYS Workbench – Modal Analysis, Geometry section.

In Mechanical, 3D Meshing has been performed with Hexa meshing done as much as possible and quad mesh in the remaining palces. The meshing was done by keeping the element size as 3mm and the Element Order as Quadratic.

After Meshing the number of nodes and elements are:

Statistics	
Nodes	79624
Elements	52295

The Mesh Qualitiy Stastistics are:

Mesh Metric	Element Quality
Min	4.6053e-002
Max	0.99991
Average	0.80885
Standard Deviation	0.13588

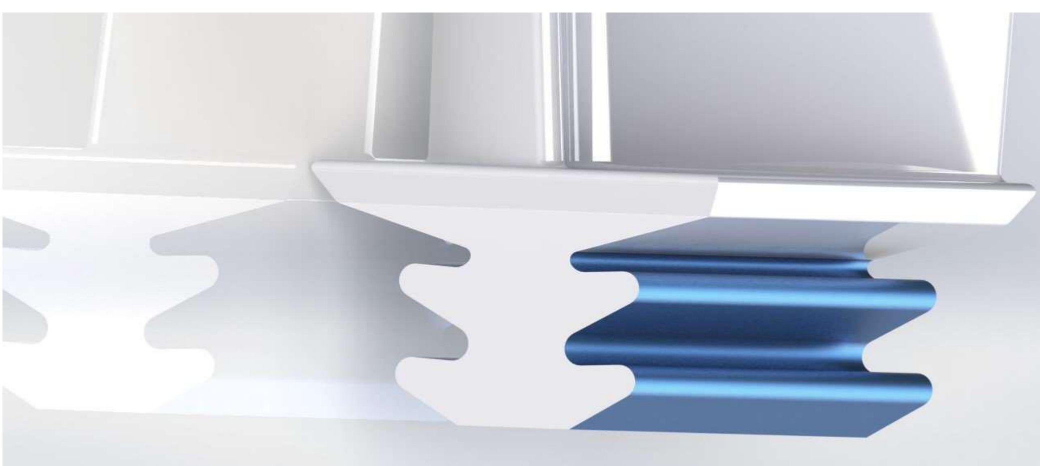
Mesh Metric	Aspect Ratio
Min	1.1671
Max	43.296
Average	2.108
Standard Deviation	2.0682

Mesh Metric	Skewness
Min	8.5885e-004
Max	0.99992
Average	0.2654
Standard Deviation	0.16735

Two structural boundary conditions namely displacement and force were applied on the rotor blade model.

The solution part of ANSYS was opened and the bottom part the blade's displacement was constrained.

U_x U_y $U_z = 0$. and Rot_x Rot_y $Rot_z = 0$.



The constraint is on both sides of the part visible in the image.

After performing Modal Analysis, the Natural Frequency of the Gas Turbine Blade are:

Tabular Data

	Mode	Frequency [Hz]
1	1.	801.17
2	2.	1874.3
3	3.	2664.
4	4.	4626.7
5	5.	6745.3
6	6.	8266.1
7	7.	8581.2
8	8.	9507.6
9	9.	10685
10	10.	12274

TITANIUM ALLOY

Table 6.3

Tabular Data

	Mode	Frequency [Hz]
1	1.	873.63
2	2.	2061.
3	3.	2981.4
4	4.	5111.
5	5.	7496.1
6	6.	9172.4
7	7.	9438.4
8	8.	10592
9	9.	11570
10	10.	13765

INCONEL 738

Table 6.4

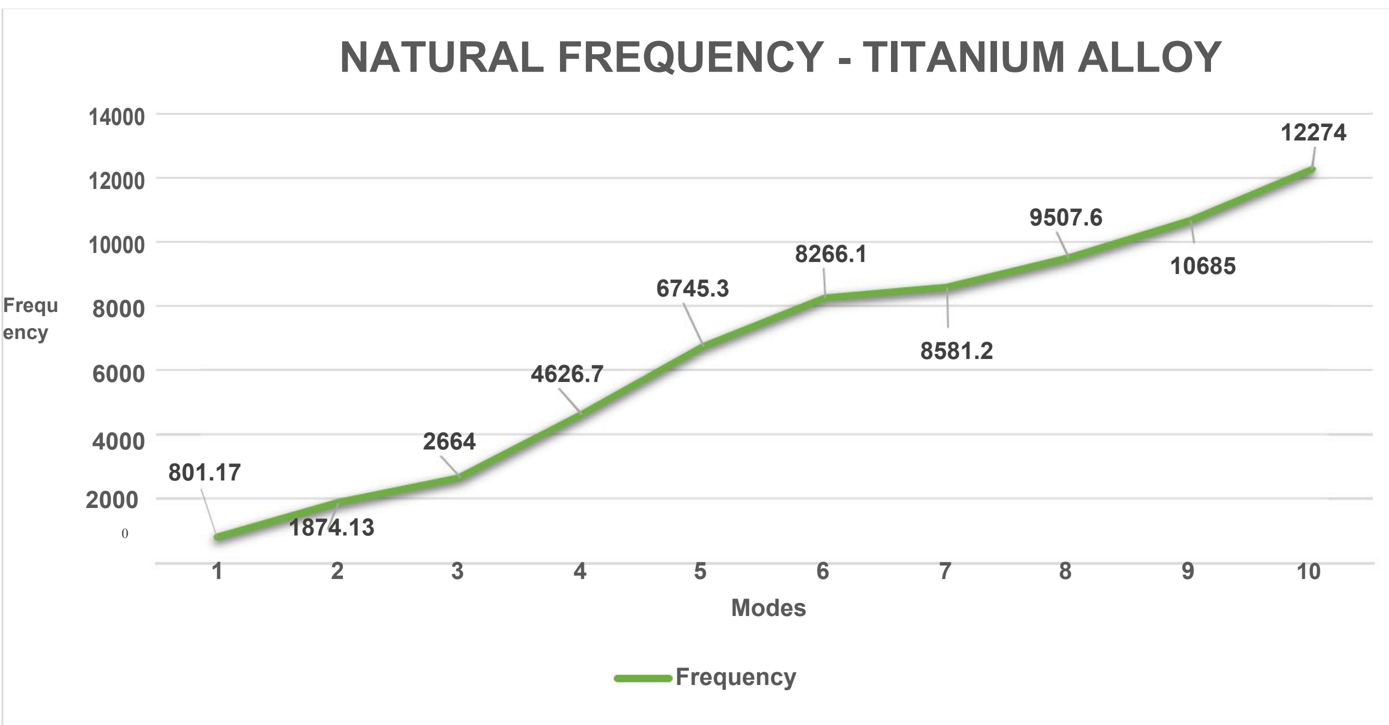
Tabular Data

	Mode	Frequency [Hz]
1	1.	882.34
2	2.	2078.1
3	3.	2995.
4	4.	5148.8
5	5.	7541.6
6	6.	9234.4
7	7.	9513.8
8	8.	10651
9	9.	11703
10	10.	13821

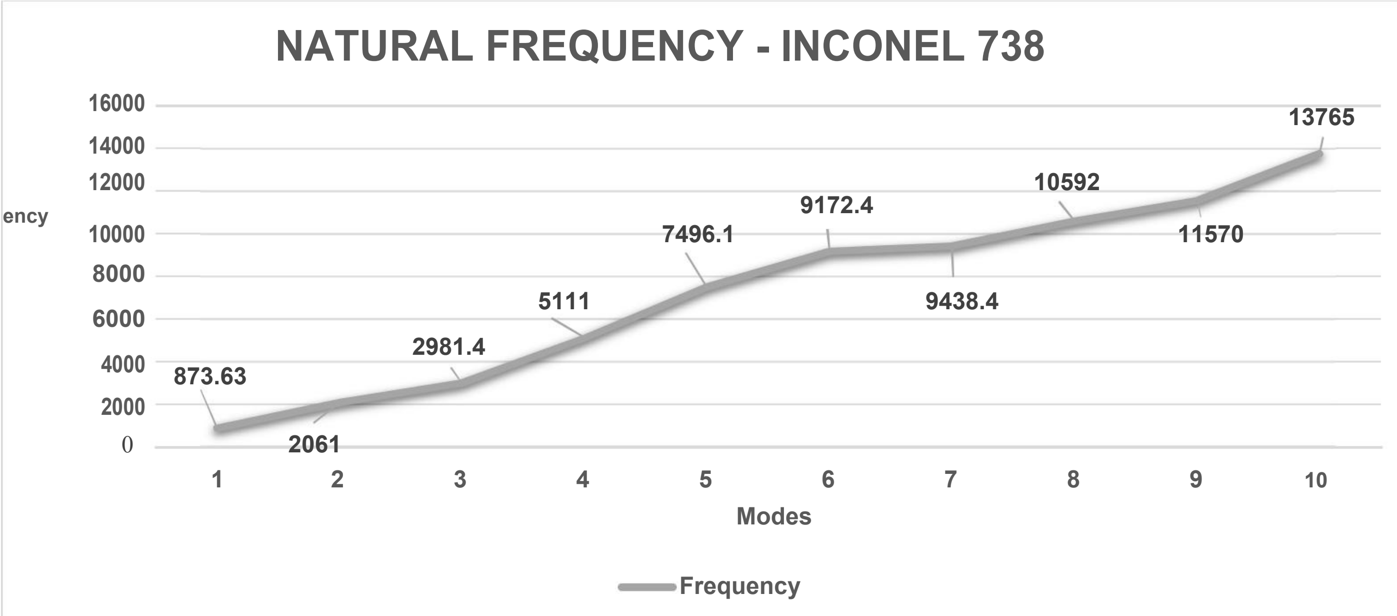
STRUCTURAL STEEL

Table 6.5

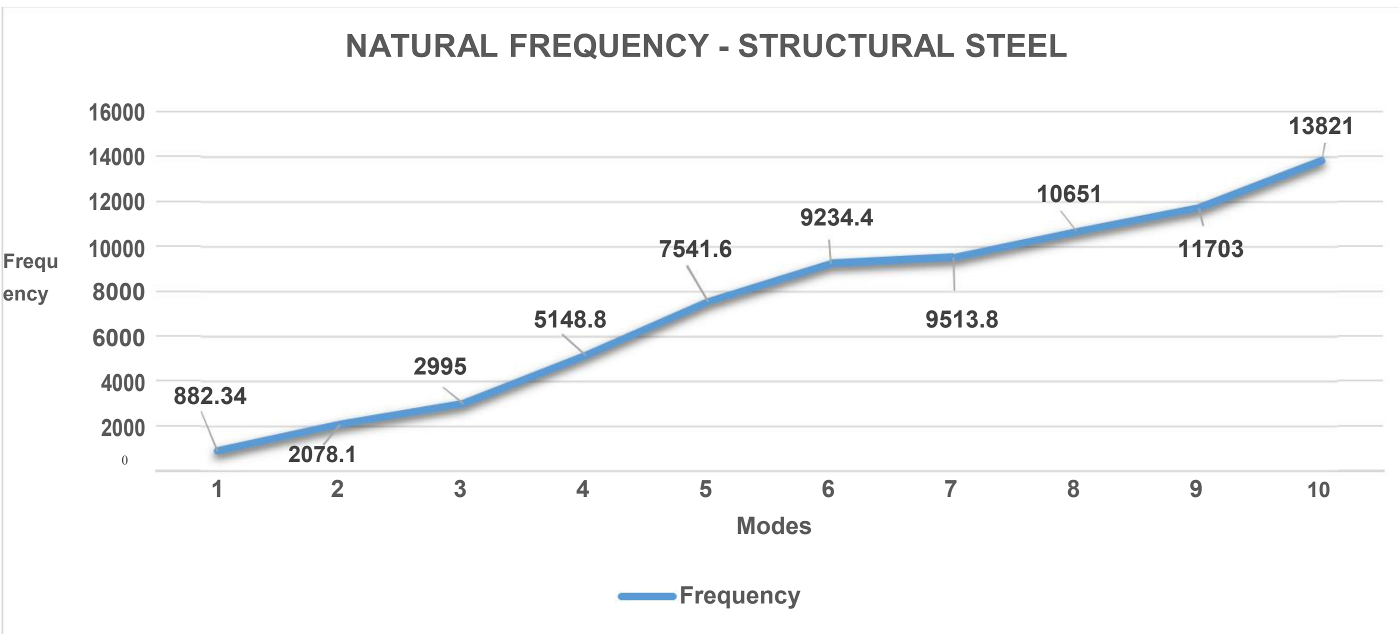
The natural frequencies for 10 modes for three materials are given below:



Graph 6.6: First 10 Natural Frequencies for Titanium Alloy.



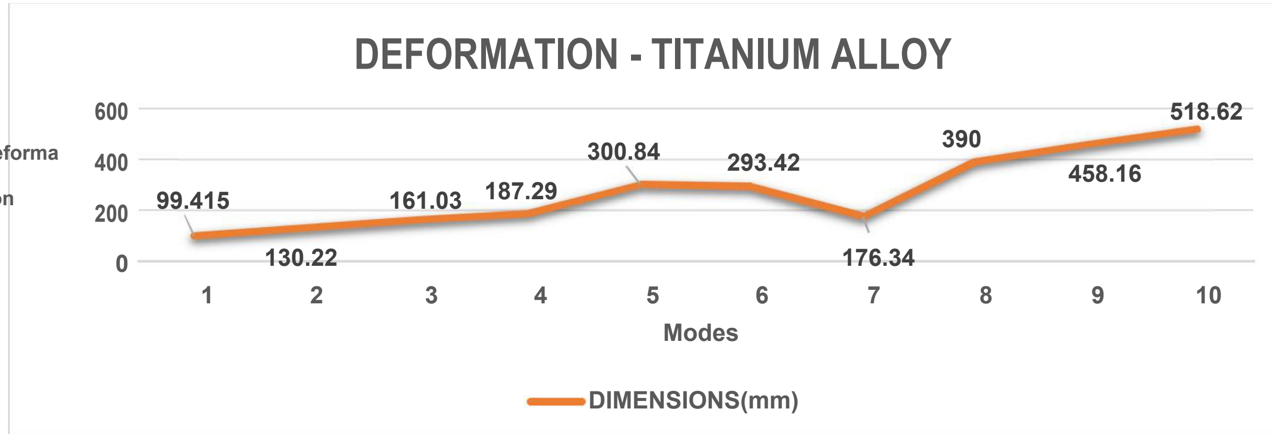
Graph 6.7: First 10 Natural Frequencies for Inconel 738.



Graph 6.8: First 10 Natural Frequencies for Structural Steel.

Deformations due to Vibrations are:

TITANIUM ALLOY



Graph 6.9: Deformations from 10 modes for Titanium Alloy

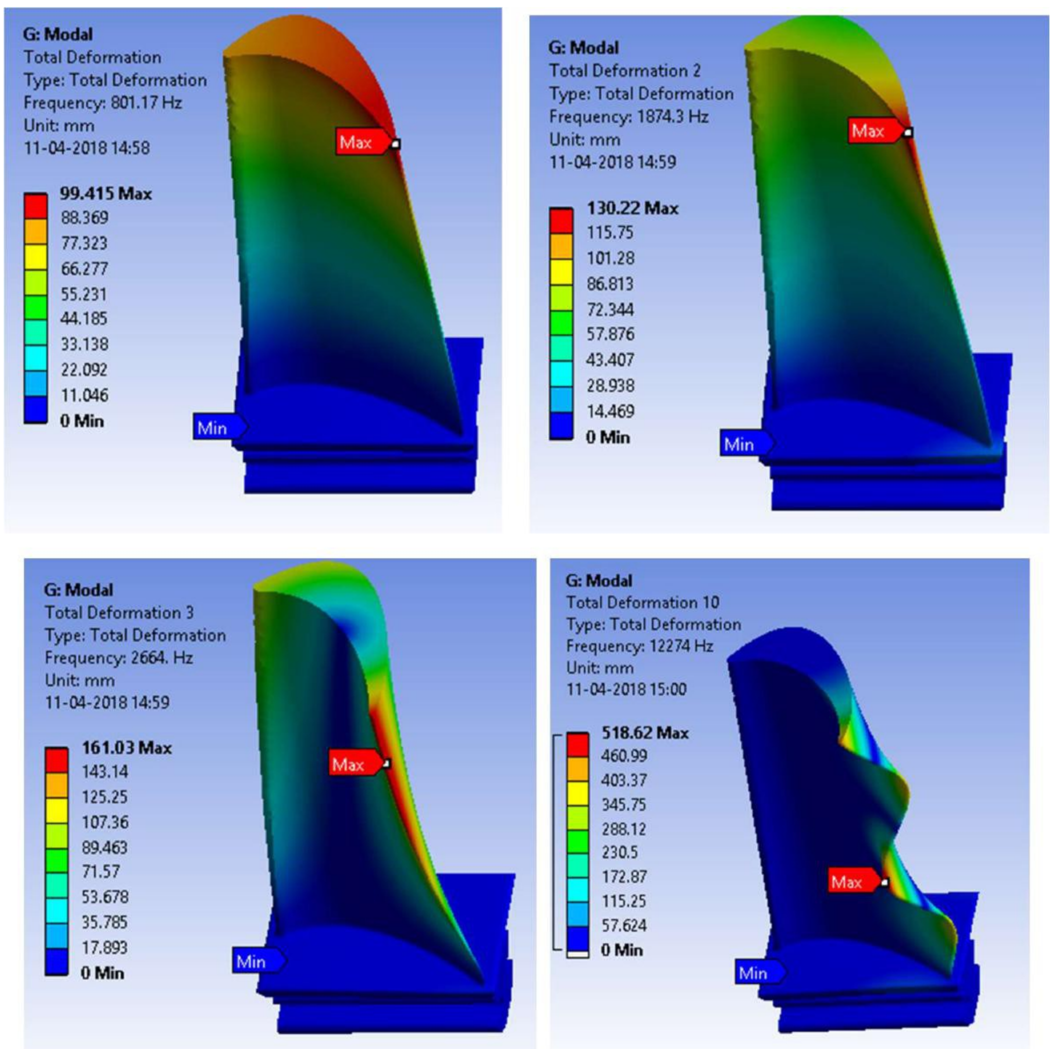
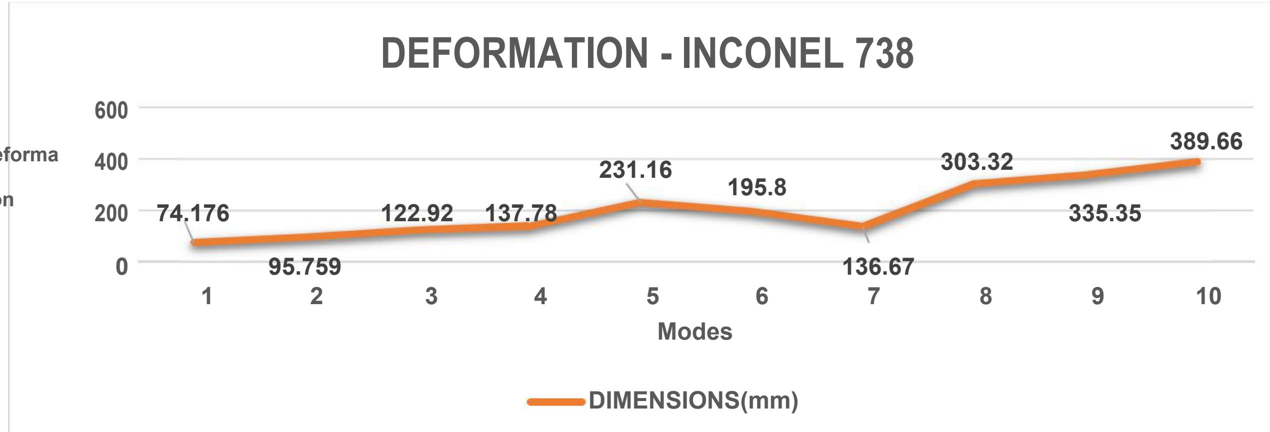


Fig 6.31 Deformations due to Natural Frequencies for Titanium Alloy

INCONEL 738



Graph 6.10: Deformations from 10 modes for Inconel 738.

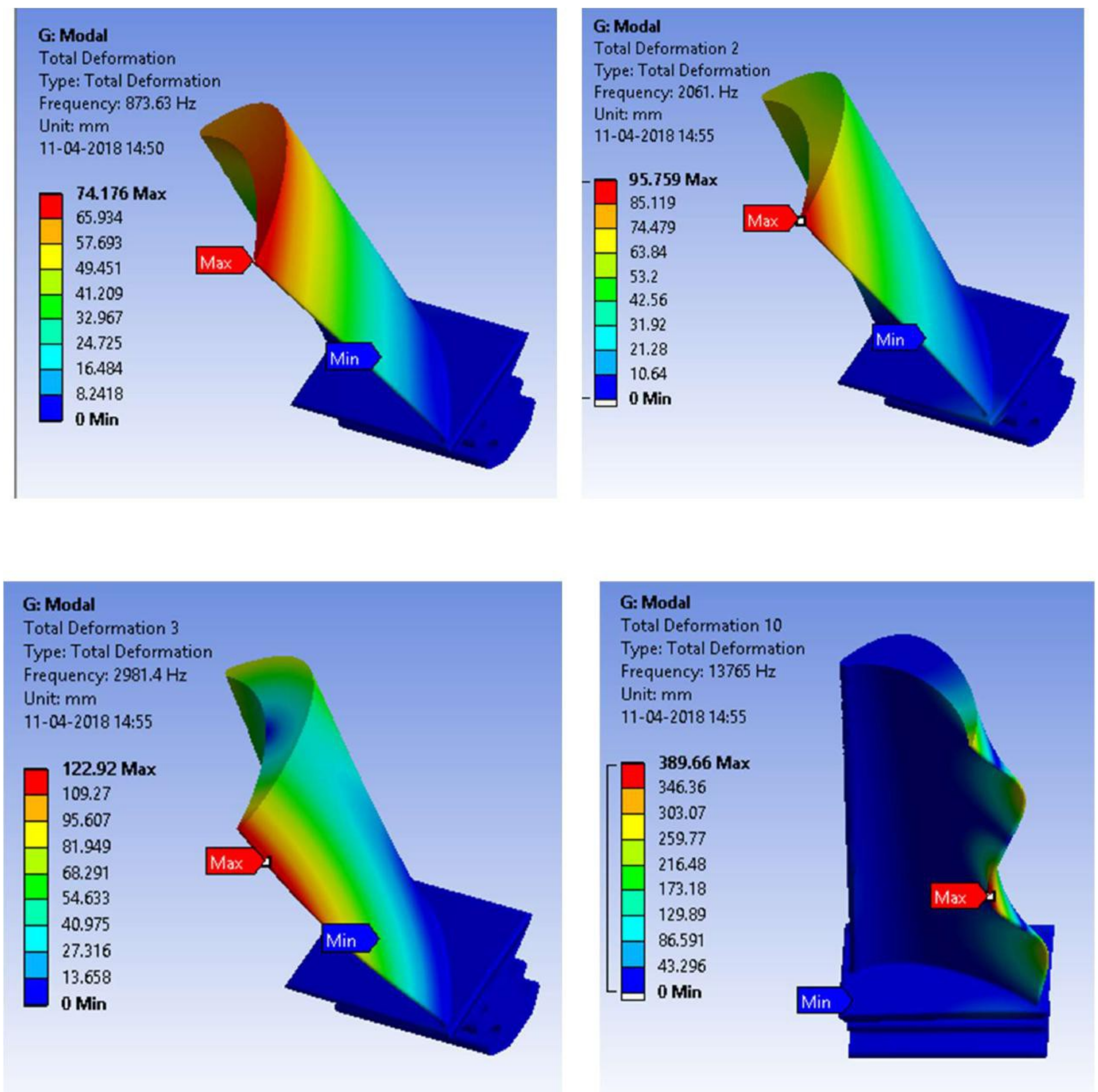
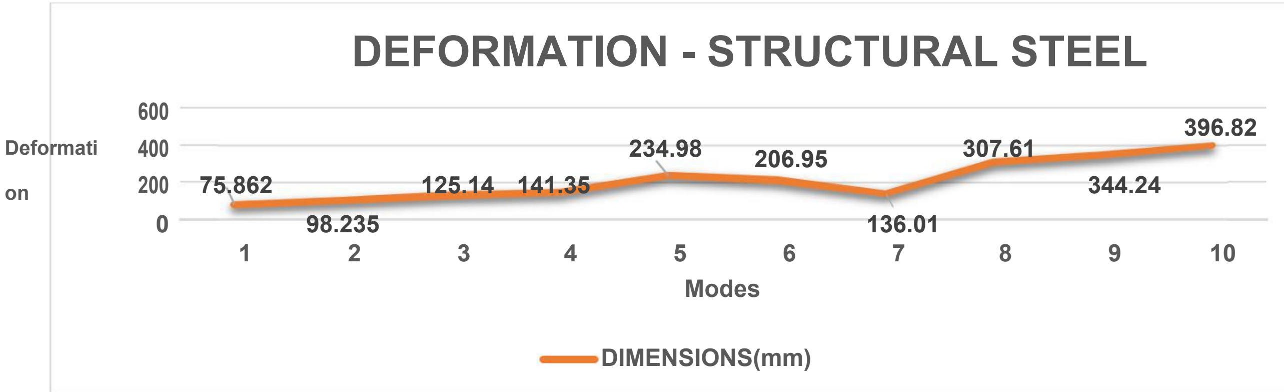


Fig 6.32 Deformations due to Natural Frequencies for Inconel 738.

STRUCTURAL STEEL



Graph 6.11 Deformations from 10 modes for Structural Steel.

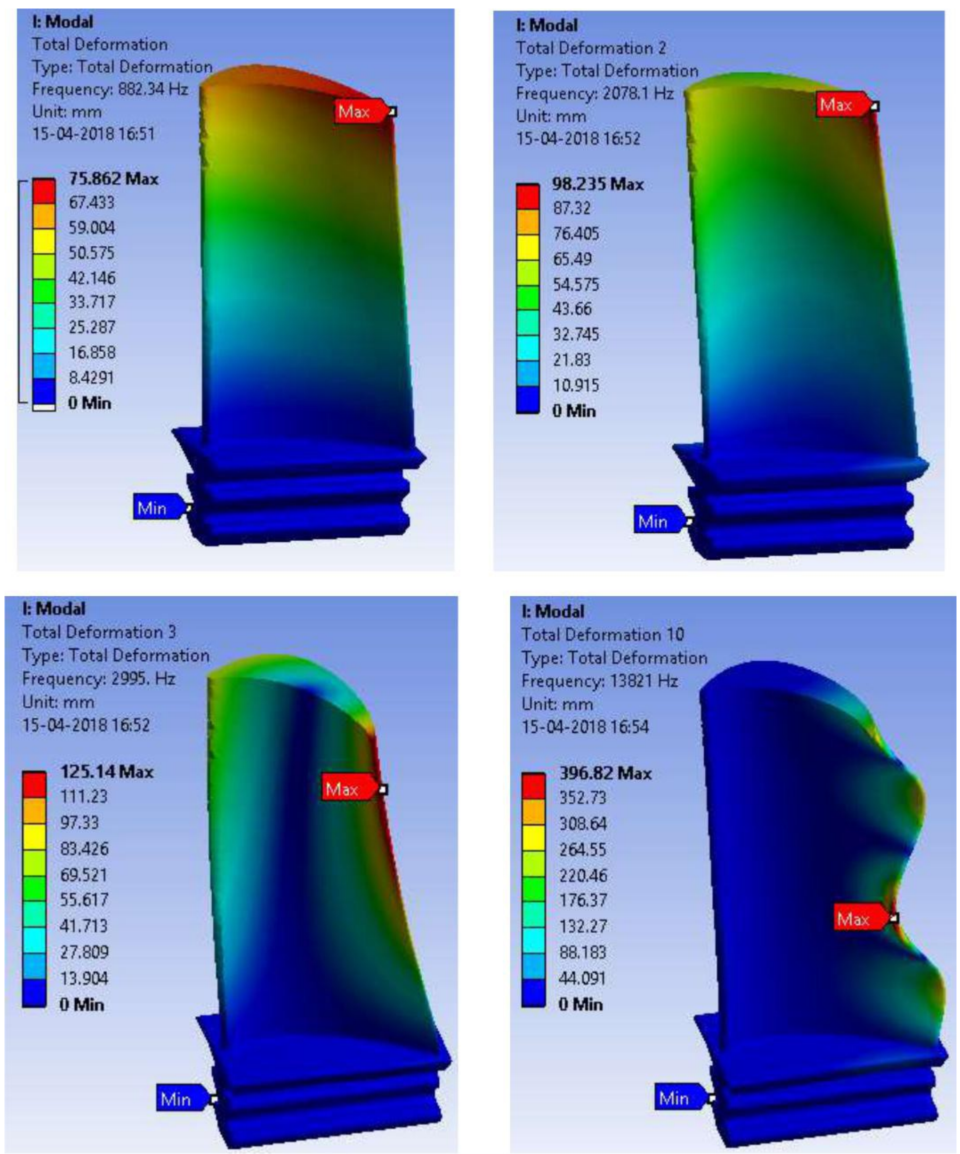


Fig 6.33 Deformations due to Natural Frequencies for Structural Steel.

6.2.4 HARMONIC RESPONSE ANALYSIS

Harmonic analyses are used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time, thus enabling you to verify whether or not your designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

This analysis technique calculates only the steady-state, forced vibrations of a structure. The transient vibrations, which occur at the beginning of the excitation, are not accounted for in a harmonic analysis.

In this analysis all loads as well as the structure's response vary sinusoidally at the same frequency. A typical harmonic analysis will calculate the response of the structure to cyclic loads over a frequency range (a sine sweep) and obtain a graph of some response quantity (usually displacements) versus frequency. "Peak" responses are then identified from graphs of response vs. frequency.

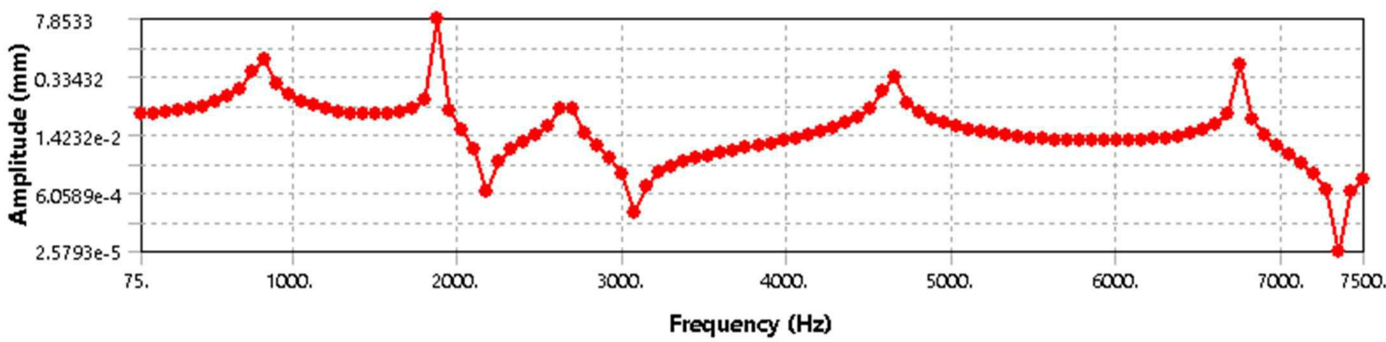
Bring Harmonic Response Analysis to the Project Schematic Section and drop it onto the Solution bar of the Modal analysis, to take the output of the modal analysis (natural frequencies) as the input to the harmonic analysis. Also apply force in the Harmonic Analysis, same as applied in Static structural, this force is the harmonic or sinusoidally.

In results we get the Amplitude vs Frequency and Phase Angle vs Frequency.

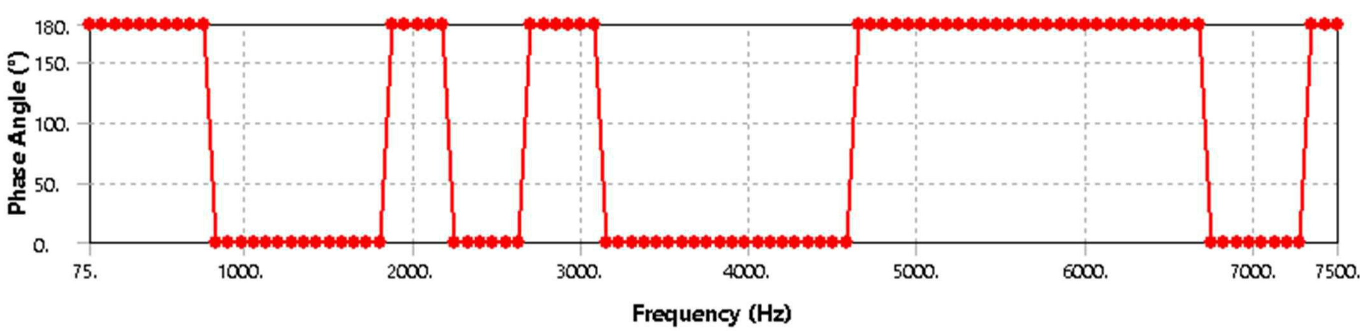
TITANIUM ALLOY

Results	
Maximum Amplitude	7.8533 mm
Frequency	1875. Hz
Phase Angle	180. °
Real	-7.8533 mm
Imaginary	0. mm

Fig 6.34 Results of Harmonic Response for Titanium Alloy



Graph 6.12 Frequency vs Amplitude for Titanium Alloy.

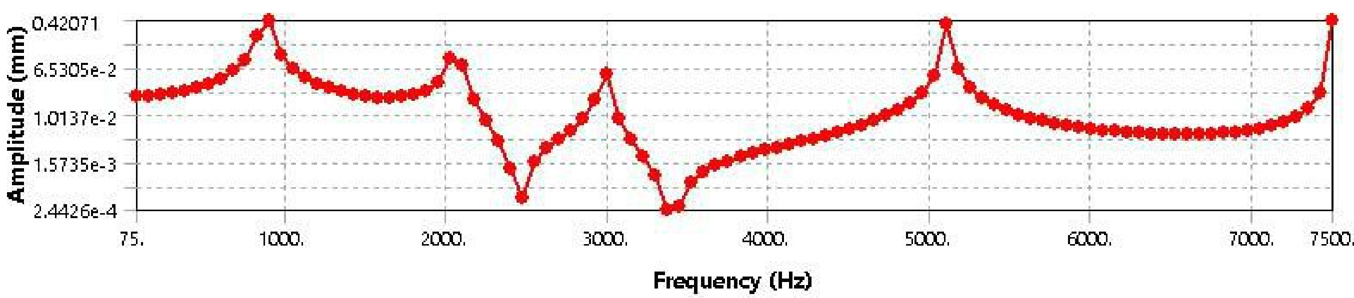


Graph 6.13 Frequency vs Phase Angle for Titanium Alloy.

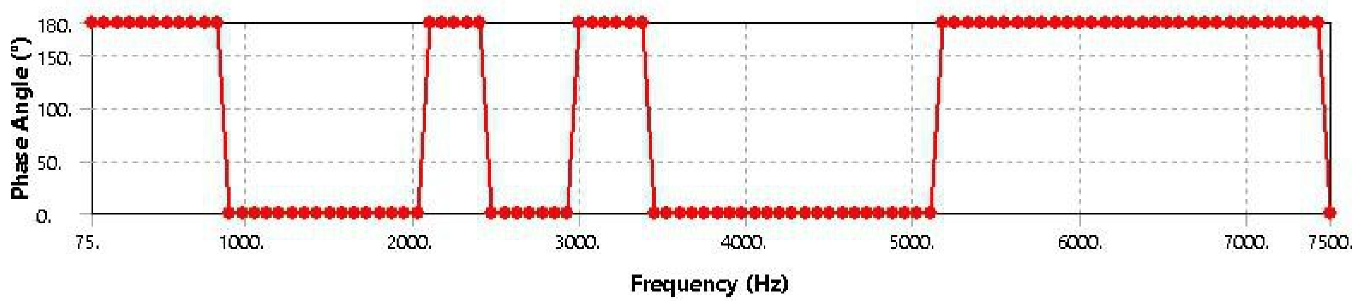
INCONEL 738

Results	
Maximum Amplitude	0.42071 mm
Frequency	900. Hz
Phase Angle	0. °
Real	0.42071 mm
Imaginary	0. mm

Fig 6.35 Results of Harmonic Response for Inconel 738.



Graph 6.14 Frequency vs Amplitude for Inconel 738.

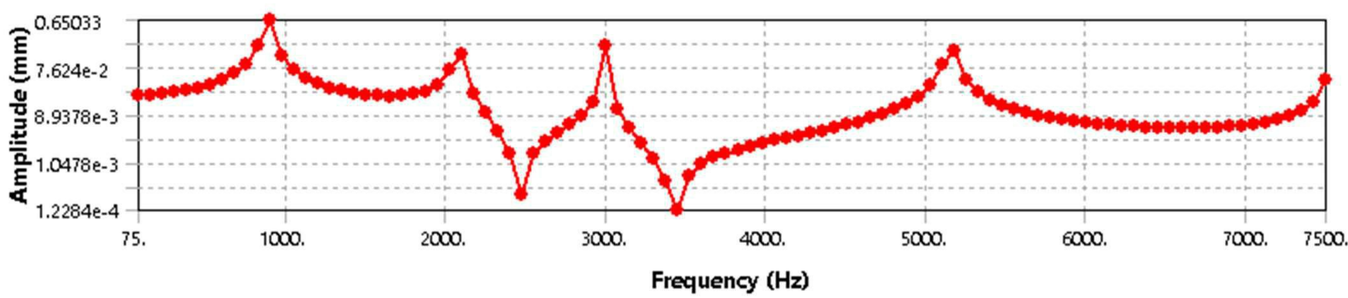


Graph 6.15 Frequency vs Phase Angle for Inconel 738.

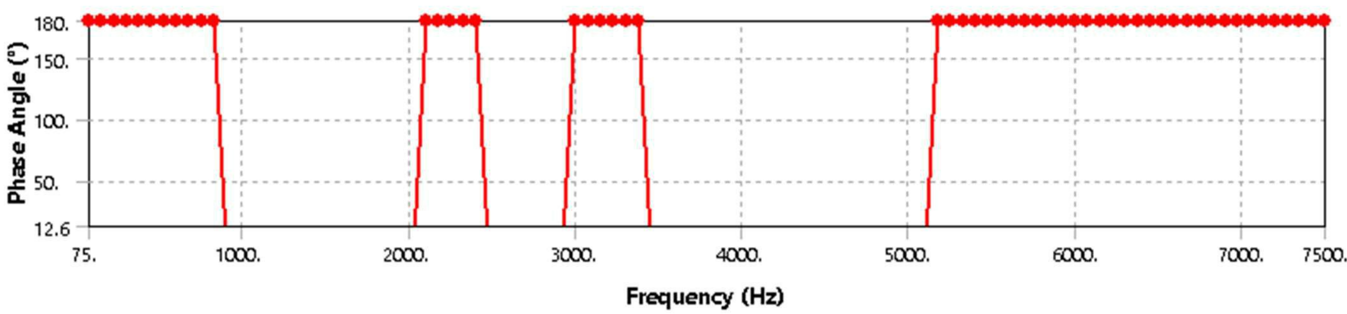
STRUCTURAL STEEL

Results	
Maximum Ampli...	0.65033 mm
Frequency	900. Hz
Phase Angle	0. °
Real	0.65033 mm
Imaginary	0. mm

Fig 6.36 Results of Harmonic Response for Structural Steel.



Graph 6.16 Frequency vs Amplitude for Structural Steel.



Graph 6.17 Frequency vs Phase Angle for Structural Steel.

CHAPTER 7:

RESULTS AND CONCLUSION

7.1 RESULTS AND DISCUSSIONS

The finite element analysis for structural, thermal, modal and harmonic response analysis of gas turbine rotor blade is carried out using ANSYS 18.1. The temperature has a significant effect on the Overall turbine blades. Maximum elongations and Temperature of gas turbine rotor blade are observed at the tip section and minimum elongation and temperature variation at the root of the blade. The structural analysis shows that the variation of stress and strain for different materials along with the deformation for the three materials. The deformations are obtained as shown in the Graph (6.1), and it is observed that the maximum deformation is 0.81395 mm, 0.664215 mm and 0.66784 mm for Titanium alloy, Inconel 738 and Structural Steel respectively. By comparing the above results the minimum deformation is to Inconel 738 and but the maximum Factor of Safety of 2.815 is for Titanium Alloy compared to 1.4089 for Inconel 738 and a bad FOS of below 1, i.e. 0.42071 for Structural Steel. This shows that Titanium alloy is safer with high FOS and deformations below safe regions. The mass of total blade is 0.366Kgs, 0.64956Kgs and 0.6225Kgs for Titanium Alloy, Inconel 738 and Structural Steel respectively. Hence for the structural stability and better design with less weight Titanium Alloy can be preferred, but is very costly so is rarely used in Gas Turbine Blades, hence the next best alternative is the Inconel 738.

The variation in the temperature is plotted and by analysing the plots of the three different materials, the Inconel 738, shows high withstanding in temperature around 806.21^{o}C, which can withstand without causing any damage to the blade. We observed the temperature variation from the tip at leading edge to trailing edge of blade the temperature is gradually decreasing and also we observed the maximum value temperature at the tip section of the blade is gradually decreasing to the root of the blade where always maximum curvature occurs and the temperature variation is less at the root of the blade. The temperature variation along X-direction is varying on front side. Inside and on the backside of the blade. The temperature decreases gradually along Z-direction. There is very small temperature gradient is occurring along Y-direction at the blade leading edge, on the suction side of blade and at the exits side of the blade.

The modal and harmonic response analysis gives almost same results giving natural frequency nearly same, which are not near to the working frequency of the turbine blade. Hence the design of the blade is safe from resonance.

7.2 CONCLUSION

The thermal stress analysis together with the mechanical stress analysis will yield more valuable information about the actual magnitudes of the Overall stresses encountered in turbine blades. In this research work we analysed and make comparing with many materials

1- The temperature has a significant effect on the overall stresses in the turbine blades.

2 - Maximum elongations and temperatures arc observed at the blade tip section 0.81395 mm, 806.21ºC and minimum elongation at the root and temperature variations at the root and the trailing edge of the blade.

3- Temperature distribution is almost uniform at the maximum curvature region along blade profile.

4- Maximum stress induced is within safe limit.

5- Maximum thermal stresses are set up when the temperature difference is Maximum from outside to inside.

6- Maximum stresses and strains are observed at the root of the turbine blade and upper surface along the blade roots.

7- Elongations in X-direction are observed only at the blade region along the blade length and elongation in Y-direction are gradually varying from different sections along the rotor axis.

8 - It could be concluded that these contour maps and profiles enables us to ascertain the areas of rotor blades that are vulnerable for failure

9 - By comparing the other two materials the stress capability for Inconel 738 it is 587 MPa and for Structural Steel it is 594.23 MPa so the material Titaniurn Alloy gives better results 330.35 MPa.

12 – If cost of the materials is not a primary issue we can select the titanium alloy which have lesser stress, lesser value

REFERENCES

1. S. S. Rao, "The Finite Element Method in Engineering", BH Publications New Delhi, 5th Edition

2, O.C. Zeinkiewicz, "The Finite Element Method in Engineering Science", Tata McGraw Hill, 2nd Edition.

3. T.R. Chandrupatla, Belegundu A.D., "Finite Element Engineering", Prentice Hall of India Ltd

4. O.P. Gupta," Finite and Boundary element method in Engineering", Oxford and IBH Publishing Company Pvt Ltd. New Delhi.

5. Principles of Turbomachinery, D.G. Shepherd

6 A Trastise on Turbomachines Dr Gopalkrishna and Dr D. Prithvi Raj

7. V. Ramamurti, "Computer Aided Design in Mechanical Engineering" Tata McGraw Hill Publishing Company Ltd. New Delhi

8. C.S. Krishnamoorthy, "Finite Element Analysis, Theory and Programming", 2nd edition, Tata McGraw Hill Publishing Company Ltd. New Delhi

9. P. RavinderReddy, "CADA Course Book", AICTE-ISTE sponsored program.

10. R. Yadav, "Steam and Gas turbine", Central Publishing House, Allahabad.

11. ANSYS Inc., U.S.A." Help Manual", ANSYS 18.1

12. P.V. Krishnakanth, G Nara Raj, R D V. Prasad, R. Saisrinu International Journal of Scientific Research Engineering & Technology (LISRET) Volume 2 Issue2 pp 060-065 May 2013 ISSN 2278 — 0882 STRUCTURE & THERMAL ANALYSIS OF GAS TURBINE BLADE BY USING F.E.M

13. Sagar P.Kauthalkar, Mr.Devendra S. Shikarwar, Dr. Pushpendra Kumar Sharma International Journal Of Engineering,Education And Technolo gy (ARDIJEET) ISSN 2320-883X. ANALYSIS OF THERMAL STRESSES DISTRIBUTION PATTERN ON GAS TURBINE BLADE USING ANSYS.

14. G.Narendranath & S.Suresh International Journal of Engineering Research and Application(IJERA) IISN: 2248-9622 Vol. 2 Issue 3.

THERMAL ANALYSIS OF GAS TURBINE ROTOR BLADE BY USING ANSYS.

15. V.Veeraragavan, International Journal of Scientific Research Engineering and Technology (IJSRET).

16. Theju V, Uday P S, PLV Gopinath Reddy, C.J.Manjunath, International Journal of Innovative Research in Science, Engineering and Technology ISSN – 2319-8753 – Vol. 3, Issue 6, June 2014.

DESIGN AND ANALYSIS OF GAS TURBINE BLADE.

17. Ch. V Prithvi Raj, K. Arun Kumar, International Journal of Science and Research (IJSR), Volume 3 Issue 3, March 2014

STRESS ANALYSIS OF GAS TURBINE MULTI STAGE ROTOR ASSEMBLY.

18. Ganta Nagaraju I, Venkata Ramesh Mamilla, M.V. Mallikarjun, International Journal of Engineering, Business and Enterprise Applications (IJEBEA) 5(1), June-August, 2013

DESIGN OPTISATION AND STATIC & THERMAL ANALYSIS OF GAS TURBINE BLADE.

19. Gujadel and M B Bhambere, International Journal of Mechanical Engineering and Robotics Research ISSN: 2278-0149 Vol. 3, No. 2, April 2014

REVIEW OF STRUCTURAL AND THERMAL ANALYSIS OF GAS TURBINE BLADE.

20. Josin George, International Journal of Latest Trends in Engineering and Technology (ILTET) Vol. 3 Issue 4, March 2014 ISSN 2278-621X

THE DESIGN AND ANALYSIS OF FIRST STAGE GAS TURBINE BLADE WITH A MODIFICATION OF COOLING PASSAGES USING ANSYS

21. V.Raga Deepu & RP.Kumar, Ropicharla International Journal of Mathematics and Engineering 170(2012) 1603-1612

DESIGN AND COUPLED FIELD ANALYSIS OF FIRST STAGE GAS TURBINE ROTOR BLADES.

22. Kamlesh Bachkar, W.S. Rathod, IJEDR, Vol. 2 Issue 2, ISSN: 2321-9939

STATIC AND THERMAL ANALYSIS OF TURBINE BLADE OF TURBOCHARGER.

23. V. NagaBhushana Rao, L N. Niranjan Kumar, N. Madhulata and A. Abhijeet, International Journal of Advance Science and Technology (IJAST), Vol.68, 2014,

MECHANICAL ANALYSIS OF FIRST STAGE MARINE GAS TURBINE BLADE.