# Dynamic Simulation of Gas Turbine Blade using Finite Element Analysis

*Shivkumar Biradar*
*Assistant Professor, Mechanical Engineering, BKIT, Bhalki. INDIA*

*(Corresponding author: Shivkumar Biradar)*

**ABSTRACT**: **From the Statistics collected for the increased evidence of Aeroengine component failure, the major reason is the High Cycle Fatigue (HCF) failure. The HCF caused by high frequency vibrations at relatively large number of cycles, where in the dominant component of strain induced during cyclic loading is elastic. Thus it is essential to design the Gas Turbine Rotor blade during design stage to avoid HCF failures. During Preliminary design stage, the HCF failure can be avoided by avoiding the resonance. The resonance occours when the natural frequency coincides with the forcing frequency giving rise to large amplitude of vibratory stresses. The natural frequencies and mode shapes can be determined from the Modal analysis of the component. Due to complicated shape of the blade, it is not possible to calculate the natural frequencies of gas turbine blade using analytical methods. Hence commercially available finite element package ANSYS is used to calculate the natural frequencies and mode shapes due to dynamic loading conditions. These frequencies are validated with the Experimental data. The Aeroengine Rotor blades are subjected to centrifugal loading conditions. Thus the centrifugal loading plays major role on the natural frequencies of the component. The component also subjected to prestressed conditions, which has an effect on the natural frequencies of the component. Hence an attempt is made to study the effect of centrifugal loading with various speeds like 0, 5000, 10000 RPM and operating speed (16043 RPM) by using the finite element analysis package such as ANSYS. Life of the gas turbine can be increased by decreasing its potential resonance effect at critical frequencies. In this project, NASA Rotor 67 blade model have been studied with variable speeds. The respective Campbell diagram shows the blade critical frequency, which needs to be reduced below the operating frequencies in order to avoid resonance and to increase the life of the turbine blades.**

**Key words:** Gas Turbine Rotor blade, HCF failure, Modal Analysis and Campbell diagram.

## I. INTRODUCTION

The basic concepts of turbine prime mover refer back to the first known turbine by Hero of Alexandria in 130 BC. However, until 1935 a practical gas turbine has not been produced that could compete with any degree of success with the reciprocating internal combustion engines and the steam plant in the fields of propulsion and stationary power plants. For many years steam is used as the working substance, and steam turbines have been developed to a high degree of efficiency. But the production of high pressure and high temperature steam requires the use of a bulky and costly steam generator. The gases produced in the steam generator never reach the turbine and they are only used to convert water into steam. Therefore if the gases can be used in the turbine the costly and wasteful intermediate step of producing steam could be eliminated. This idea resulted in the development of a gas turbine.

Even though the gas turbine has a long history, John Barber did not make the first important design until 1791 in England. The next important development was accomplished in the early part of the nineteenth century by Sterling and Ericson who developed hot air engines that led to the establishment of the sterling constant volume cycle and Ericson constant pressure cycle, both using isothermal compression and expansion. In the middle of the nineteenth century, Joule proposed the constant pressure cycle using isentropic compression and expansion. This cycle is now used as the fundamental cycle for gas turbine.

**Working principle of Gas Turbines:** The basic simple open cycle gas turbine consists of a compressor, combustion chamber and a turbine. The compressor takes in ambient air and raises its pressure. In the combustion chamber heat is added to the air, raising its temperature and hence its heat energy.

The product from combustion chamber then called working fluid is expanded through a turbine to develop mechanical energy.
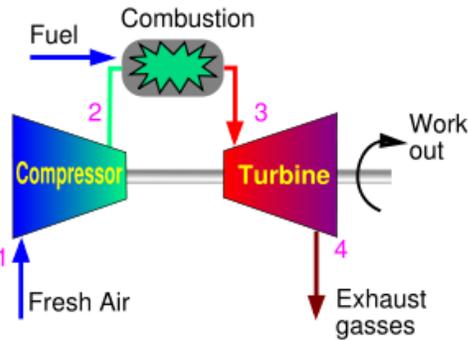


**Fig. 1.** Gas Turbine Open Cycle.

**Gas Turbine Cycle:** The analysis of the gas turbine process is based on Brayton cycle is called ideal cycle which has dry air with variable specific heats as the working fluid.

The Brayton cycle is shown in Fig.1. Atmospheric air is compressed isentropic ally to several times its atmospheric pressure in the compressor, which enters combustion chamber by injecting and burning the fuel in the air stream. Since a normal hydrocarbon fuel burns with a temperature around 1930 °C to 2200 °C, the major part of the air, called secondary air is utilized to cool the products of combustion down to the permissible turbine inlet temperature. The combustion in the combustion chamber is at constant pressure. These hot and high-pressure gases from the turbine are let out into the atmosphere. Approximately two thirds of the power developed by the turbine is utilized to drive the compressor and the auxiliaries and about one third of the power of the turbine goes for useful work. A fresh charge of air is now taken into compressor to begin the next cycle. Therefore in the Brayton cycle, the cycle is completed through the atmosphere and heat on supplied by directly injecting the fuel in the air stream. Brayton cycle is an open cycle where as the Joule cycle is a closed cycle. In the Joule cycle the same working substance i.e., air is circulated and recirculated. Heat exchange is by indirect means in the heat exchangers. Except for these differences the Brayton cycle is identical to Joule cycle in all aspects. Practical gas turbines use Brayton cycle. Losses occur in all components. Therefore the power output for a given amount of fuel supplied depends upon the efficiency of the components. Further, power output of the gas turbine plant depends upon the amount of fuel supplied. The temperature in turn is limited by the availability of materials for the construction of a turbine blade and the life required. Therefore the two factors that determine the performance of a gas turbine are component efficiencies and maximum cycle temperature.

In a gas turbine, fuel must be added not only to increase the energy of the working media to produce the necessary work but also to make up for losses which are radiation, leakage, pressure drops and efficiencies of component parts. Thus, the problem becomes one of the largest yields for least fuel consumed.

The Thermal efficiency of an actual simple open cycle gas turbine is heavily dependent upon the following operating variables:

1. Pressure Ratio and Turbine Inlet Temperature
2. Turbine Efficiency
3. Compressor Efficiency
4. Atmospheric or Compressor Inlet Air Temperature.

This thesis is mainly based on turbine blade materials and its life of the components so by consider the turbine blades and its performance

**Turbine blades:** These are one of the most important components in a gas turbine power plant. These are the components across which flow of high pressure gases takes place to produce work. A blade can be defined as the medium of transfer of energy from gases to the turbine rotor. The shape or profile of a blade in the turbo machine cascade is of an airfoil shape. The airfoil profile is designed using airfoil theory.

**Typical Gas Turbine:** Fig. 2 represents the Typical Turbofan Gas Turbine engine. It has mainly 4 sections Compressor, Combustor, Turbine and Accessory. The compressor and Turbine Rotors are identified as critical parts as they are subjected to heavy Thermo Mechanical
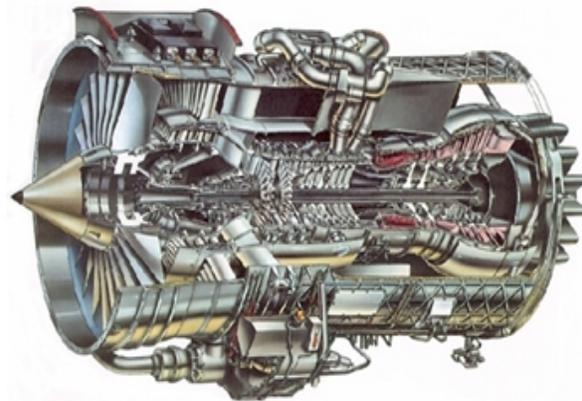


**Fig. 2.** 3D view of Typical Aeroengine.

Since the Aeroengine components are subjected to High temperature environment, the material chosen should have high strength to weight ratio along with the capability to withstand its properties at high temperature. The Rotors are made of Titanium alloy.
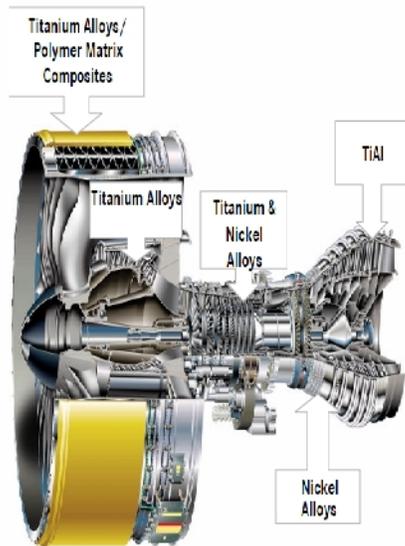


**Fig. 3.** Typical materials used in Aeroengine parts.

**Structural Mechanical assessment of Turbines:** The process of structural Mechanical assessment of a Turbine rotor component starts from the design mission which is usually defined in the Engine Specification. The design mission provides the required thrust and power as a function of time. It is assumed that at this stage the part geometry and the materials are already defined. In a first step, there are engine performance parameters derived from thrust and power requirements. The performance parameters consist of the temperatures and pressures in the main gas path and the spool speeds for each point of the design mission. Additionally, the cooling air flows, temperatures and pressures in the secondary air paths are determined. In a second step, these performance and cooling system parameters are used as boundary conditions for the calculation of transient temperature distributions in the engine components. The third step is concerned with the mechanical analysis. Total stresses are calculated as sum of centrifugal stresses (due to part rotation), thermal stresses (induced by temperature gradients), and stresses from pressures and of assembly stresses (fits and bolt clamping). Based on the results of the stress analysis, the critical areas of the rotor can be identified. Critical areas are those areas which are exposed to the highest stresses and largest stress ranges,

and which can be expected to determine the fatigue life of the part. Thus Fatigue is an important failure mode in an Aeroengine. High Cycle Fatigue (HCF) and Low Cycle Fatigue (LCF) are the major drivers for failure in an Aeroengine. In this project we are mainly focusing the High Cycle Fatigue that occurs due to high frequency vibrations at relatively large number of cycles.

Following are the Causes for the HCF in a typical Aeroengine:

**1)** Mechanical vibration arising from Rotor imbalance
**2)** Aerodynamic excitation occurring in upstream vanes, downstream struts and blades whereby engine excitation and component response frequencies corresponding to different modes of vibration may overlap.

Current design methods to assess the HCF life of critical components in aircraft gas turbine engines entail following general steps:

**1)** Appropriate stress analysis using FE analysis are performed to determine mean stress level.
**2)** Structural Dynamics FE simulations are carried out to determine resonant frequencies and Excitation modes.
**3)** The design of component is then carried out in such a way that it meets: a) criteria for safe life for HCF with the appropriate mean stress level (using modified Goodman diagram).b) No potential resonance related problems arise.

Mechanical design mainly includes Stress analysis and Modal analysis. Thus it is very essential to carry out the modal analysis in the preliminary design stage to determine the dynamic characteristics of the Structure. Figure indicates the overview of mechanical design process in an Aeroengine.

order to meet the HCF criteria of design, frequency criteria needs to be met. The frequency criteria can be met during design phase by plotting the Campbell diagram to verify the potential resonance. The Campbell diagram is a plot of natural frequency versus speed along with the forcing functions. Modal analysis needs to be carried out at the operating speed of the engine to extract the natural frequencies.

## II. LITERATURE REVIEW

Rotating shafts are employed in industrial machines such as steam and gas turbines, turbo generators, internal combustion engines, reciprocating and centrifugal compressors for power

transmission. On account of the ever increasing demand for power and high speed transmission, the rotors of these machines are made extremely flexible, which makes the study of vibratory motion an essential part of design. The shafts of these machines are subjected to torsional and bending vibrations and in some cases unstable conditions of operation.

According to M.L.J.Verhees [1], the main cause of turbine blade failure is high cycle fatigue. Fatigue failure is related to repeated cyclic loading on a structural member. The fatigue life of a structural member i.e. the number of load cycles it can survive is in general determined by the magnitude of the stress cycles. The exact relation between the magnitude of the stress and the fatigue life depends on the material properties of the structural member. In this research M.L.J.Verhees considered the turbine blade as a Euiler-Bernouli beam and he find out the modal analysis for the turbine blade by considering the clamped and unclamped ends and finally he compared the results with Euiler-Bernouli beam. In principle there are two ways in which the failure of turbine blades due to fatigue problems can be eliminated. Those are a correct structural design and the prevention of cyclic loading. The correct design of a structural member can usually eliminate or dramatically reduce fatigue problems. For a turbine blade this is not always possible. The design of a blade is usually constrained by aerodynamic properties, weight, rotor length, etc. which can make the elimination of fatigue problems through design modifications.

According to J.Hou and B.Wicks [2], turbine Blades have been traditionally modeled as a cantilever supported beam in his research blade root flexibility represented by using springs attached to one end of the beam. In these assumptions the analytically intermediate interactions between the blade and disks fit-trees are simplified in order to achieve a linear vibration system. According to his research, vibration is generally recognized as one of the most significant causes of high cycle fatigue failure in gas turbine engines. The blade root flexibility often cannot be determined easily in the assessment of blade vibration behavior and exclusion of this effect may lead to a false prediction of vibration characteristics. His report outlines a study of root flexibility and aerofoil untwists effects on the vibration characteristics of a turbine blade using 3-D finite element analysis, together with laboratory testing. The vibration characteristics of the blade were analyzed and the predicted results were correlated with laboratory test results at ambient temperature. The vibration characteristics of the blade assembly during engine operation were then predicted through a pre-stressed modal analysis, including the

centrifugal stiffening effect, the temperature effect and the interactions between the blade and disc. The root flexibility effect due to differences in clamping condition was then investigated in detail. Aerofoil untwist effects due to creep were also revealed in this analysis.

The effect of blade root flexibility for a turbine blade has been investigated Using advanced FE techniques, including a pre-stressed and pre-creped modal analysis for the blade. In order to make reliable predictions for various loading conditions, the FE model was validated with experimental test results and close agreement was achieved for all the available test cases. Thermal and centrifugal loads have inherent effects on the vibration characteristics of the blade. For a simple clamped condition, the natural frequency tends to increase with the centrifugal stiffening effect and decrease with the thermal effect, leading to an overall decrease in the natural frequency when the two effects are combined.

According to Achuta Rio [3], Rotor dynamics is a collective term for the study of vibration of rotating parts found in a wide range of equipment including turbines, power stations, machine tools, automobiles, home appliances, aircraft, marine propulsion systems, medical equipment and more. In these applications, resonant vibration in which mechanical systems can oscillate excessively when excited by harmonic loads at their natural frequencies is of particular concern. These large-amplitude vibrations can bend and twist rotating shafts, leading to premature fatigue failure in these components as well as bearings and support structures. Also, deformation of shafts and other components can cause rotating systems to impact adjacent parts in which clearances are tight, causing potentially catastrophic damage in high-speed equipment. Analysis of rotating systems typically involves the study of many different variables related to vibration including the critical rotational speeds that set up natural-frequency resonances, the response of the entire system to unbalanced loads and instabilities, deflection of the shaft during vibration, torsional vibration in which shafts also twist around their axes, and flow-induced oscillations produced by fluids moving through the system. Calculation of these and other vibration-related variables can be performed in ANSYS Mechanical software using some of the most advanced rotor dynamics simulation capabilities available in commercial finite element analysis (FEA) codes. Rotor dynamics usually is best studied in the rotating frame of reference, in which Coriolis terms are used in the equations of motion to describe rotational velocities and accelerations.

Introducing these Coriolis terms for static, modal, harmonic and transient analysis provides a modified equation of motion: [M], [C] and [K] are the structural mass, damping and stiffness matrices, respectively. [Kc] is the spin softening matrix, and [G] is a "damping" matrix contribution due to the rotation of the structure or the Coriolis term. This modified equation of motion is at the foundation of performing the most common types of rotor dynamics analyses.

## III. PROBLEM DEFINITION

**Scope for the present work:** The scope for the present work includes analytical assessment of the dynamic characteristics of the Gas Turbine Rotor. The dynamic characteristics include the Eigen values (Natural frequencies) and the Eigen Vectors (Mode shapes). The analytical assessment is done by using the Commercial FE package ANSYS software. The Dynamic Response of the Rotor subjected to centrifugal loading is computed. In the present work the Geometry used for analysis purposes corresponds to NASA Rotor 67. The Geometry is created using the NASA design code generated co-ordinates. The geometry created is validated with the Actual Geometry of the NASA Rotor with the experimental data. The validation is done by comparing analytical frequencies obtained with the experimental data of the NASA Rotor. The Dynamic Analysis is carried out considering the prestressed effect due to centrifugal loading. Analysis is also carried out without the prestressed effect. The effect of response with and without pre-stress effect is being studied to verify the impact on Campbell diagram. The analysis is carried out at various speeds to see the effect on natural frequencies and on the resonance condition. The Dynamic analysis is carried out at following RPM's:

1) 0 RPM, Room temperature
2) 5000 RPM
3) 10000 RPM
4) 16043 – Operating speed of the Rotor.

Finally a Campbell diagram is plotted to see the impact on the resonance due to various speeds with and without the prestressed effect. The Frequencies and mode shapes are provided for all the cases of analysis. The Campbell diagram represents the plot of speed versus frequency along with the forcing frequencies. The forcing frequencies are the excitation frequencies due to rotor unbalance, upstream and downstream vanes. The upstream and downstream excitation depends on the number of the vane count on the upstream and down stream of the Rotor. The resonance is identified at the crossing point of the forcing frequencies and the natural frequencies.

The upstream and downstream excitation can be explained with the help of an example. Suppose a particular rotor stage is having 7 stator vane count on the upside and 14 stator vane count on the downside of rotor, the excitation for the Rotor is 7th Harmonic and 14th Harmonic. The 1st Harmonic is the fundamental frequency and the higher harmonics are the multiples of these frequencies. These forcing harmonics vary depending on the surrounding vane count. It depends on the Rotor stage considered for the analysis.

**Geometric modeling of the NASA Rotor 67:** The NASA Rotor 67 is a transonic fan. The first stage of a NASA Lewis designed two-stage axial-flow fan was used as the Geometry in this project.

The rotor consists of 22 low aspect ratio blades and is designed to produce a total pressure ratio of 1.63 for a mass flow rate of 32.25kg/sec with a tip inlet relative Mach number of 1.38 at the design point of speed 16,043 rpm which gives design tip speed of 429 m/sec. The aspect ratio for these blades is around 1.56. The rotor inlet and exit tip diameters are 51.1 and 49.5 cm respectively, and the rotor inlet and exit hub/tip radius ratios are 0.375 and 0.475 respectively. The running rotor tip clearance is 0.5 mm. Fig.3.2 represents the Rotor Blade manufacturing co-ordinates system used by the NASA design code. Fig.3.3 represents the Rotor blade sections at three span wise locations. Table.AII.1 represents the Rotor blade manufacturing co-ordinates generated by the NASA design code.

The Geometry is created in Ansys V11.0. The local co-ordinate system is created at each of the Rotor sections. The Blade manufacturing co-ordinate system is used as a reference system at each of the cross sections. The co-ordinates are read into Ansys V11.0 and then rotated with the proper given stacking angle. Using these co-ordinates splines are generated in Ansys at each of the cross sections. Using these splines volumes are built to create the required Rotor blade profile. Table.AII.2 represents the Ansys generated blade co-ordinates used for the creation of blade.
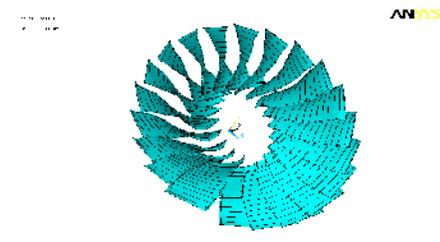


**Fig. 4.**

**FE modeling of the Rotor:** Since all the Rotors are assumed to behave in a similar manner one rotor is considered for the Dynamic analysis. One rotor corresponds to 360/22 Sector. Since the blade Hub is rigid portion the hub is not modeled in the present analysis. Also we are focusing the frequencies of the blade. The Geometry created in Ansys is meshed with the Linear Hexahedron elements. The element types used for this purpose are Solid 45. The mesh size used for the model is around 0.038". SOLID45 is used for the 3-D modeling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. While meshing the Geometry a 2D regular hexa mesh is prepared using which the 3D elements are being generated. The 2D elements used for this purpose are MESH 200 elements that have no effect during finite element computation.
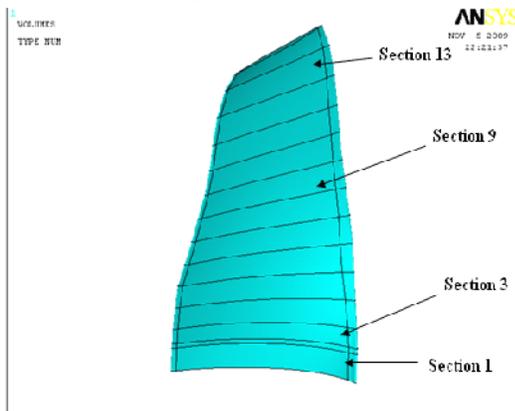


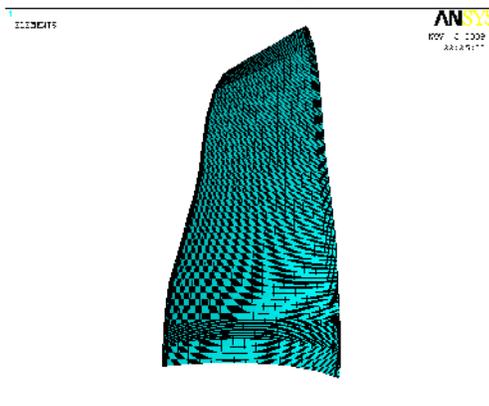**Fig. 5.** Sectional volumes of the blade geometry before meshing.



**Fig. 6.** Represents the FE model of the blade.

## IV. SOFTWARE DESCRIPTION AND ANALYSIS PROCEDURE

**Overview of ANSYS:** ANSYS is a general-purpose finite element-modeling package for numerically solving a wide variety of mechanical problems. These problems include: Static/Dynamic Structural Analysis (both linear and non-linear), Heat Transfer and Fluid Problems, as well as Acoustic and Electro-Magnetic problems.

In general, a finite element solution may be broken into the following three stages. This is a general guideline that can be used for setting up any finite element analysis.

- Preprocessing: defining the problem

The major steps in preprocessing are given below:

- Define key points/lines/areas/volumes
- Define element type and material/geometric properties
- Mesh lines/areas/volumes as required

The amount of detail required will depend on the dimensionality of the analysis (i.e. 1D, 2D, axis-symmetric, 3D).

- Solution: assigning loads, constraints and solving
Here we specify the loads (point or pressure), constraints (transnational and rotational) and finally solve the resulting set of equations.

- Post processing: further processing and viewing of the results: In this stage one may wish to see
- Lists of nodal displacements
- Element forces and moments
- Deflection plots
- Stress contour diagrams

**WHAT IS ANSYS?** ANSYS IS FINITE ELEMENT ANALYSIS SOFTWARE THAT ENABLES ENGINEERS TO PERFORM THE FOLLOWING TASKS.

- Build computer models or transfer CAD models of structures, products, components, or systems.
- Apply operating loads or other design performance conditions.
- Study physical responses, such as stress levels, temperature distributions, or electromagnetic fields.
- Optimize a design early in the development process to reduce production costs.
-

- Do prototype testing in environments where it otherwise would be undesirable or impossible (for example, biomedical applications).

The ANSYS program has a comprehensive graphical user interface (GUI) that gives users easy, interactive access to program functions, commands, documentation, and reference material. An intuitive menu system helps users navigate through the ANSYS program. Users can input data using a mouse, a keyboard, or a combination of both.

**ANSYS Basics:** Element name: The ANSYS element library consists of more than 100 different element formulations or types. An element type is identified by a name (8 characters max), such as BEAM4, consisting of a group label (BEAM) and an identifying number.

Real constants: Data which are required for the calculation of element matrix, but which cannot be determined from the node locations or material properties, are input as "real constants". Typical real constants include area, thickness, inner diameter, etc.

Material properties: Various material properties are used for each element type. Typical material properties include young's modulus, density, co-efficient of thermal expansion, etc. Some of them are called linear properties, which require single iteration. Properties such as stress-strain data called non-linear because it requires an iterative solution.

Surface loads: Various element types allow surface loads. They are typically pressures for structural element types, etc. Body loads: Various element types allow body loads. They are typically pressures for structural element types, etc.

Natural coordinate system: This is the local system, which permits the specification of a point within the element by a dimensionless numbers whose magnitude never exceeds unit.

Main menu: The menu contains main ANSYS functions such as preprocessing, solution, and post processing. All functions are "modal" with respect to each other i.e.: one function must be completed before starting the next.

Output window: The output window receives all the text output from the program-command like notes, warnings, errors and any other messages.

ANSYS tool bar: The toolbar is a set of push buttons that executives commonly used ANSYS functions. Some push buttons are predefined. This feature increases the productivity by giving one-button access to favorite and frequently used functions.

Graphics windows: The graphics window is where all ANSYS graphics displays are drawn and all graphical picking is done. It is the largest of the GUI windows.

**Structural Analysis:** Structural analysis is probably the most common application of the finite element method. The term structural implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical and mechanical structures such as ship hulls, aircraft bodies and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

**Types of structural analysis**

The seven types of structural analysis available in the ansys family of products are listed below. The primary unknowns calculated in a structural analysis are displacements. Other quantities, such as strains, stresses and reaction forces, are then delved from the nodal displacements.

Structural analyses are available in the ANSYS/Multiphysics, ANSYS/Mechanical, ANSYS/Structural and ANSYS/Linear plus programs only. One can perform the following types of structural analyses.

Static Analysis-Used to determine displacements, stress etc., under static loading condition both in linear and nonlinear static analyses. Non-linear ties can include plasticity, stress stiffening, large deflection, large strain, hyper-elasticity, contact surfaces and creep.

Modal Analysis- Used to calculate the natural frequencies and mode shapes of a structure.

Harmonic Analysis- used to determine the response of a structure to harmonically time varying loads.

Transient Dynamic Analysis- used to determine the response of a structure to arbitrarily time-varying loads.

Spectrum Analysis- An extension of the modal analysis, used to calculate stresses and trains due to a response spectrum or a PSD input (random violations).

Buckling analysis- used to calculate the buckling loads and determines the buckling mode shape. Both linear buckling and nonlinear buckling analysis are possible.

Explicit Dynamics Analysis- ANSYS provides an interface to the LS-DYNA explicit finite element program and is used to calculate fast solutions for large deformation dynamics and complex contact problems.

**Static analysis in FEM:** Assumptions used in linear static analysis. A number of important assumptions and limitations are inherent in linear static analysis are summarized as follows:

Linear Elastic material

**Table 1. Indicates the comparison of the experimental and the analytical frequencies.**

| MODE | Frequency (Hz) Tabulation | | % Difference in frequencies |
|---|---|---|---|
| | NASA Experimental frequencies at 16043 RPM | Ansys NASA Rotor67 at 16043 RPM | |
| 1. | 601.0 | 529 | 12 |
| 2. | 1307.9 | 1219 | 6.8 |
| 3. | 1913.2 | 1991 | 4 |
| 4. | 2740.6 | 2653 | 3.2 |
| 5. | 3148.2 | 3059 | 2.8 |

The material is assumed to be homogeneous and isotropic. We restricted to material in which stress is directly proportional to strain and to loads that not take the material beyond its permanent yield point (the material remains elastic).
Small displacement

Analysis is restricted to the small displacement assumption used in the formulation of governing equation for linear beam, shell, plate and solid behavior in ANSYS element development.
Slowly applied loads

In linear static analysis our structure is in static equilibrium. Loads must be "slowly applied", which means that they don't include dynamic effects. Some type of loads such as impact loads violates this restriction in obvious way.

**Performing a static analysis:** The procedure for a static analysis consists of the following tasks:
- Build the model
- Set solution controls
- Set additional solution options
- Apply the loads
- Solve the analysis
- Review the results

The ANSYS program offers the following approaches to model generation:
- Creating a solid model with in the ANSYS.
- Using direct generation.
- Importing a model created in computer-aided design (CAD) system.

**V. RESULTS AND DISCUSSION**

**Validation part:** The below table indicates the Comparison of Experimental frequencies with the

Dynamic analysis frequencies. The frequencies are very close to the Experimental data. Slight deviation exists because of the numerical approximation during FE analysis. Thus it concludes that the Blade Geometry and the frequencies obtained from FE analysis can be used for design evaluation purposes.
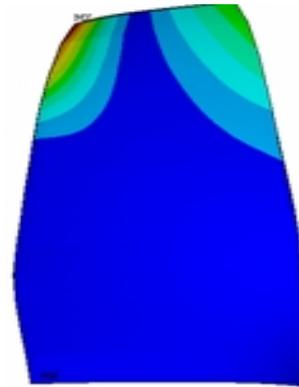


**Fig. 7. 1ˢᵗ Torsional mode shape of Ansys analysis.**

**Table 2. Frequency comparison with and without prestress conditions at various RPM'ˢ**

| Rotor speed in RPM | Mode | With prestress | Without prestress | % Difference in frequencies |
|---|---|---|---|---|
| | | Case A | Case B | |
| 5000 | 1 | 442 | 424 | 4 |
| | 2 | 1087 | 1071 | 2 |
| | 3 | 1946 | 1941 | 0.25 |
| | 4 | 2566 | 2555 | 0.43 |
| | 5 | 2990 | 2982 | 0.27 |
| 10000 | 1 | 473 | 403 | 15 |
| | 2 | 1132 | 1067 | 6 |
| | 3 | 1961 | 1938 | 1 |
| | 4 | 2595 | 2554 | 2 |
| | 5 | 3013 | 2980 | 1 |
| 16043 | 1 | 529 | 355 | 33 |
| | 2 | 1219 | 1059 | 13 |
| | 3 | 1991 | 1934 | 3 |
| | 4 | 2653 | 2553 | 4 |
| | 5 | 3059 | 2976 | 3 |

The above figure shows the comparison of the Experimental Mode shape with the FE mode shape thus they are very close. This also confirms that the FE analysis results are in good agreement with the Experimental results.

**Frequencies:**

**Table 3: Natural Frequency tabulation at room temperature.**

| Mode | 1 | 2 | 3 | 4 | 5 |
|------|---|---|---|---|---|
| At Room Temperature Without load. | 431 | 1072 | 1941 | 2556 | 2983 |

From the frequency tabulation it is observed that at the room temperature the frequencies are very low as the Rotor is not subjected any loads.
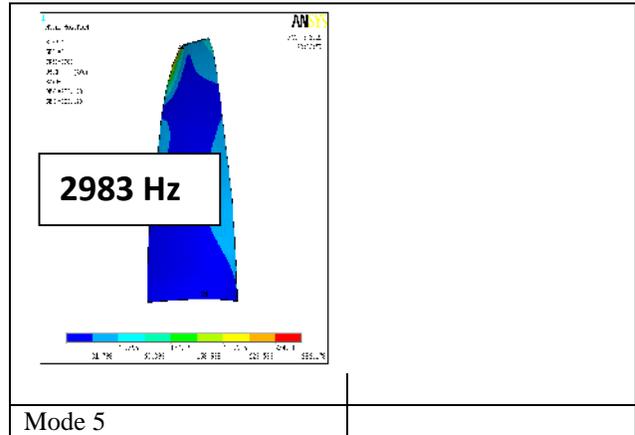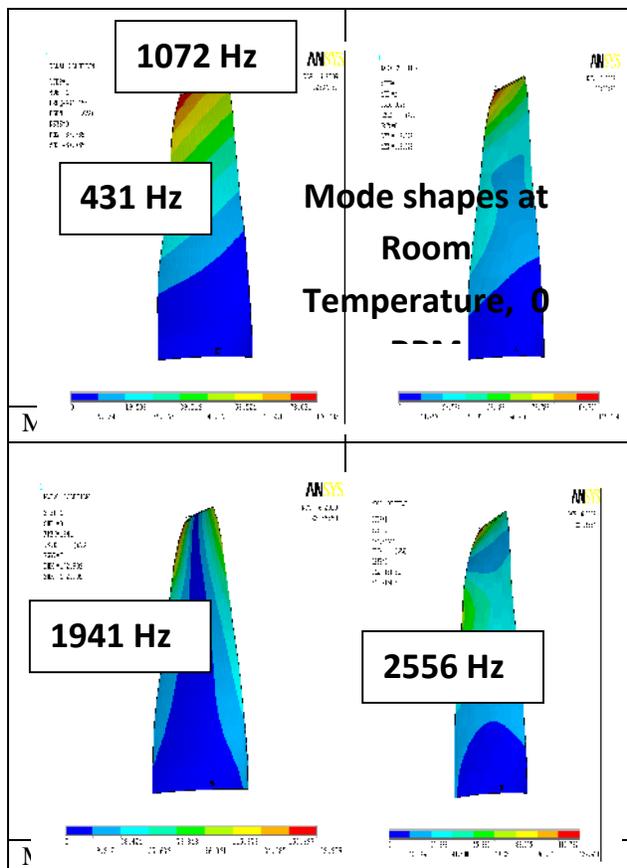
**Mode shapes:**





Mode 5

**Fig. 8.** Mode shapes at Room temperature, 0 RPM.

Based on the Modal analysis at various RPM's the mode shapes with and without prestressed effect remain same. The natural frequencies altered. The variation in natural frequencies affects the resonance.

From the Campbell Diagrams it is observed that Mode 3 has potential resonance with $7^{th}$ Harmonic (7E – marked with red color). Thus we are studying the effect on resonance due to change in RPM's as well as with and without prestress effects. With prestress effects as the speed increases the frequencies also increase. On plotting Campbell diagram for these cases at lower speeds the resonance is close to the running speed of the Engine, thus we expect to have high stresses. These stresses acting for larger cycles leading to HCF failure of the component. At higher speeds as frequencies increase the resonance will be shifted slightly away from the operating speed, resulting in lower stresses on the component. Also without the prestress effect the frequencies are lower compared to with prestress effect, thus the resonance is close to operating speed resulting in higher stress. Thus it is very essential to include the prestress effect.

## VI. CONCLUSIONS

-If the resonance is close to the operating speed range of the engine, we can tune the frequencies away from the running range of the Engine.

-It is very essential to carry pre stressed modal analysis during design evaluation to study the dynamic characteristics of the structure

-If the resonance is observed for many modes within the operating range of the Engine, it is difficult to tune all the frequencies.

Thus at this stage the mode shapes of the structure help in determining the potential failure modes that can be tuned to avoid the resonance.

-The FE frequencies are having good agreement with the experimental frequencies, the FE analysis can be powerful tool that helps in quick turnaround time, without experimentally validating the new design every time. This helps in avoiding costly experiments.

-It is observed that frequencies are very close to the Experimental data, slight deviation exists because of the numerical approximation during FE analysis. Thus it concludes that the Blade Geometry and the frequencies obtained from FE analysis can be used for design evaluation purposes

## REFERENCES

[1] Stuart Moffatt, *"Blade forced response prediction for industrial gas turbines part I: methodologies",* International Gas Turbine & Aero engine Congress & Exhibition June 16-19, 2003, Atlanta, Georgia, USA.

[2] Avinash, V.Sarlaskhar, *"Blade pro an Ansys-based turbine blade analysis system",* Impact Technologies, LLC, and Rochester, NY 14623, U.S.A.

[3] Ewins, D.J.Imregun, *"Vibration modes of packeted bladed disks"*, transactions of the ASME, Vol. 106 pp175-180.

[4] Genta, G, *"Dynamics of Rotating Systems",* Springer ISBN 978-0-387-20936-4.

[5] Rao, J. S, *"Natural frequencies of Turbine Blading – A survey"*, Shock and Vib.Dig. 5(10), P

[6] M. L. J. Verhees, *"Experimental Modal analysis of a gas turbine blade"* Technische Universite it Eindhoven.2004

[7] J. Hou and B.Wicks, *"Root flexibility and untwist effects on vibration characteristics of gas turbine blade"* Air Vehicles Division Platforms Sciences Laboratory, Defence Science & Technology DSTO-RR-0250, Australia.2002

[8] Achuta Rao, *"Rotor dynamic capabilities in ANSYS Pro-mechanical",* ANSYS. 2000

[9] Michael D. Hathaway, *"Unsteady Flows in a Single-Stage Transonic Axial-Flow Fan Stator Row"*, AVSCOM Lewis Research Center.