Industrial Science
Vol.1,Issue.6/Aug. 2014 ISSN : 2347-5420

Research Papers

FINITE ELEMENT ANALYSIS ON AXIAL COMPRESSOR BLADE

Shruthi J H[1] and Ravi Kiran B S[2]

[1]M.Tech Student, Product Design and Manufacturing,
Dept IEM, Sri Siddhartha Institute of Technology, Tumkur, , Karnataka.
[2]Assoc Professor, Department of Industrial Engineering and Management,
Sri Siddhartha Institute of Technology, Tumkur, 572105, Karnataka.

Abstract

Compressor is one of the most important part in gas turbine. The compressor has compressor blades which compress the gas which flows from the inlet and leaves the gas with higher temperature and pressure to the outlet. The project is to design and analysis of compressor blade. CATIA is used to design the solid model and ANSYS for analysis for Finite Element model generation by applying boundary condition which also includes specific post-processing and life assessment of the blade. However the program makes effective use of the ANSYS Pre-processor to mesh complex compressor blade geometries and apply boundary conditions.

The objective of the current task is to perform the Static and Dynamic (modal) analysis to know the stress and strain at the high temperature zone.

KEYWORDS:

Compressor blade, Design, CATIA, ANSYS, Static analysis, Dynamic analysis.

I.INTRODUCTION

The evolution is a balanced integration economic, aerodynamic, structural, dynamic, noise, aesthetic consideration, which are known to be machine type and size dependent. The design of a modern compressor blade includes choices of blade number, airfoils, chord and twist distributions, and materials. The justification for each of these choices often includes conflicting considerations that need to be prioritized. For example, thin airfoils are desirable for their high lift-to-drag ratios and are roughness tolerant, whereas thick airfoils sacrifice some of these qualities to achieve the greater blade stiffness required for large machines.

Many problems in engineering and applied science are governed by differential or integral equations. The solutions to these equations would provide an exact, closed form solution to the particular problem being studied. However, complexities in the geometry, properties and in the boundary conditions that are seen in most real-world problems usually means that an exact solution cannot be obtained or obtained in a reasonable amount of time. Current product design cycle times imply that engineers must obtain design solutions in a 'short' amount of time. They are content to obtain approximate solutions that can be readily obtained in a reasonable time frame, and with reasonable effort.

The Finite Element Method (FEM) is one such approximate solution technique. The FEM is a numerical procedure for obtaining approximate solutions to many of the problems encountered in engineering analysis. In the FEM, a complex region defining a continuum is discretised into simple geometric shapes called elements. The properties and the governing relationships are mathematically in terms of unknown values at specific points in the elements called nodes. An assembly process is used to link the individual elements to the given system.

When the effects of loads and boundary conditions are considered, a set of linear or nonlinear algebraic equations is usually obtained. A solution of these equations gives the approximate behavior of the system.

2. AXIAL COMPRESSOR

An axial compressor is a pressure producing machine. It is a rotating, air foil-based compressor in which the working fluid principally flows parallel to the axis of rotation. This is in contrast with other rotating compressors such as centrifugal-compressors, axial-centrifugal compressors and mixed-flow compressors where the air may enter axially but will have a significant radial component on exit. The energy level of air or gas flowing through it is increased by the action of the rotor blades which exert a torque on the fluid which is supplied by an electric motor or a steam or a gas turbine.

Axial flow compressors produce a continuous decelerating flow of compressed gas, and have the benefits of high efficiency and large mass flow rate, particularly in relation to their cross-section. They do, however, require several rows of airfoils to achieve large pressure rises making them complex and expensive relative to other designs.

3. OBJECTIVE OF THE PROJECT

1.To design the compressor blade
2.By studying the design, applying the Finite Element Modelling for the blade
3.Conduct the static and dynamic analysis to the blade.

4. FINITE ELEMENT ANALYSIS

A commercial FEA process consists of these 3 steps

Pre -processor
Processor
Post-processor

Pre-processor - Input data describes geometry, material properties, loads and boundary condition. Software can automatically prepare much of the FE mesh, but must be given direction to the type of the element and the mesh density desired. That is the analyst must select the desired formulation that suits the mathematical model and state how large the element size should be in selected portion of FE model.

Processor- Software's automatically generates matrices that describes the behavior of each element, combines these matrices into large single matrix equation that represent the FE structure, and solves this equation to determine values of field qualities of nodes. Substantial additional calculations are required if behavior is non-linear or time dependent.

Post processor - FEA solution and quantities derived from it are listed or graphically displayed. This set up is also automatic, except that the analyst must tell the software what list is displayed to prepare. In stress analysis, typical display include deformed shape, with deformations exaggerated and probably animated, and stress of various types on various plane

The below figure shows the basic general process in the analysis of any typical product.

5. ANALYSIS

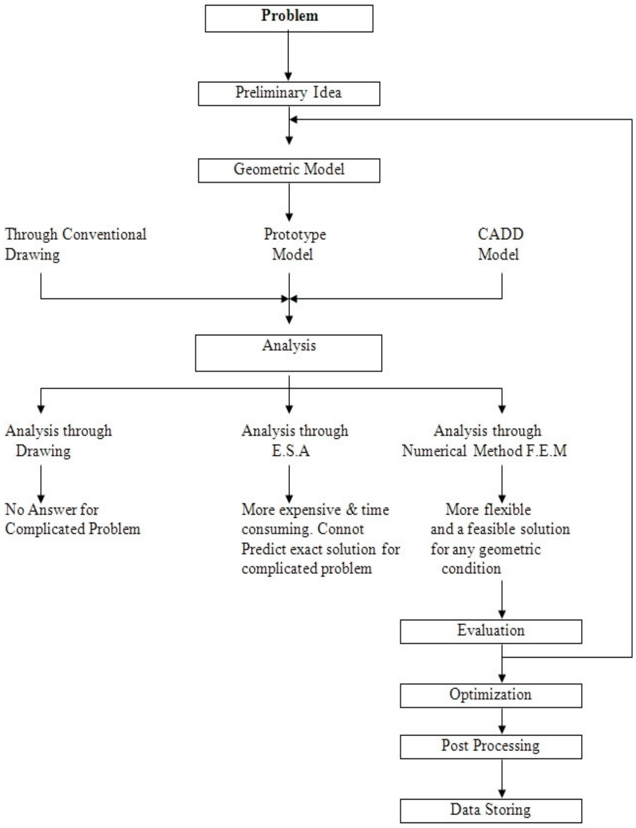

Fig 5.1 Analysis of typical product

6. FINITE ELEMENT ANALYSIS OF COMPRESSOR BLADE

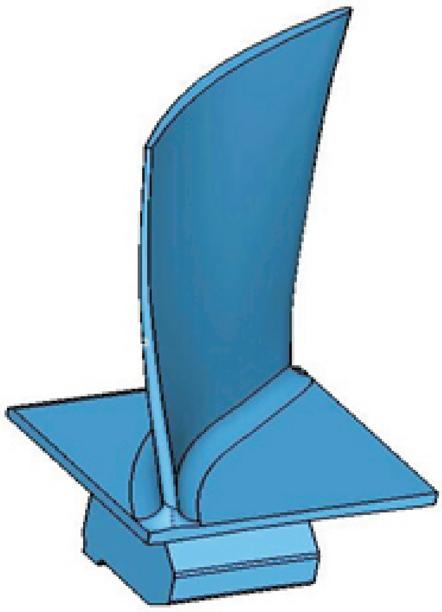

Fig 6.1: compressor blade

6.1Modelling

The modelling process is divided into two steps.

1. Geometric Modelling
2. Finite Element Modelling

6.1.1 Geometric Modelling

Geometric modelling phase is to represent the geometry in terms of points, curves, surface and volumes. The geometric model stores enough information to fully describe the boundaries, surfaces and topology of the object. During present work geometric model is constructed by using Catia V5.

6.1.2 Finite Element Modelling

Here the geometric model is used in generating Finite element model by meshing the area using the appropriate elements. The density of mesh is decided depending upon the accuracy of the result required. So the geometric model was meshed with fine elements at the constraint and at airfoil region. The geometry is meshed with tetrahedral elements.

6.2 Mesh Generation

Hyper mesh as Pre-processor & ANSYS post processing software's were used.

Meshing means dividing the component into small components or elements. The meshing is done using Hyper mesh v10 software. The meshed elements are tetrahedral in form. The reason for using tetrahedral elements is that the compressor blade is a solid component and to any solid part it is compulsory to use tetrahedral elements and also this helps in time consuming.

The problem required that the fine mesh be generated at the constraint region and at the region of interest. Convergence test for the element test was done before going to the real life calculation.

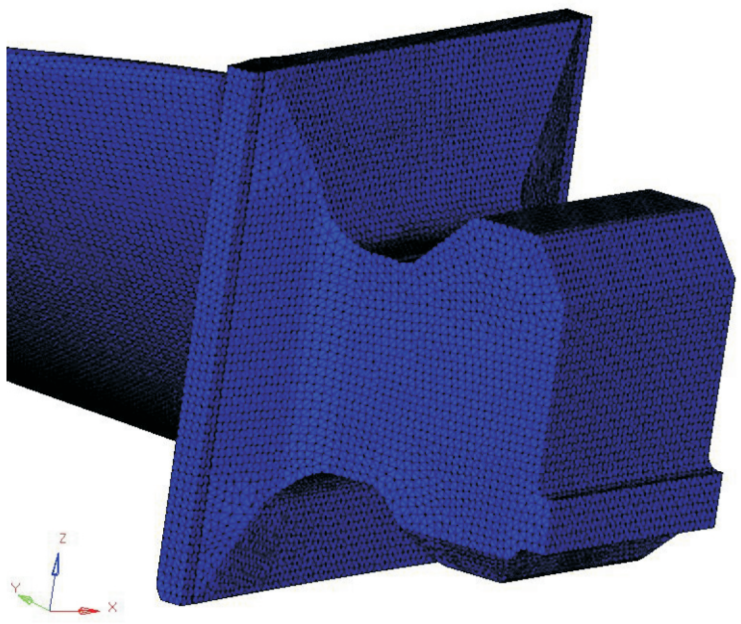

Fig 6.2: Meshed compressor blade

6.2.1 Load and Boundary Condition

The two different loads are applied to the bottom of the blade and those two types are axial loads and tangential loads.

Boundary condition is applied in all degree of freedoms to the bottom of the compressor blade.

6.3 ANALYSIS

6.3.1 Static Analysis:

A static analysis calculates the effects of steady loading condition on a structure while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads.

Static analysis is used to determine the displacements, stresses, strains and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.

The analysis is carried out using ANSYS 11. The reason behind using this software is that, the compressor blade is the place where the temperature is high when compared to other regions and ANSYS is the simulation software which gives accurate results than other related software's. In static analysis we are doing analysis for tensile stress and displacement in all (X, Y, Z) directions. And also conducting von misses stress. The flowing figures shows the analysis.

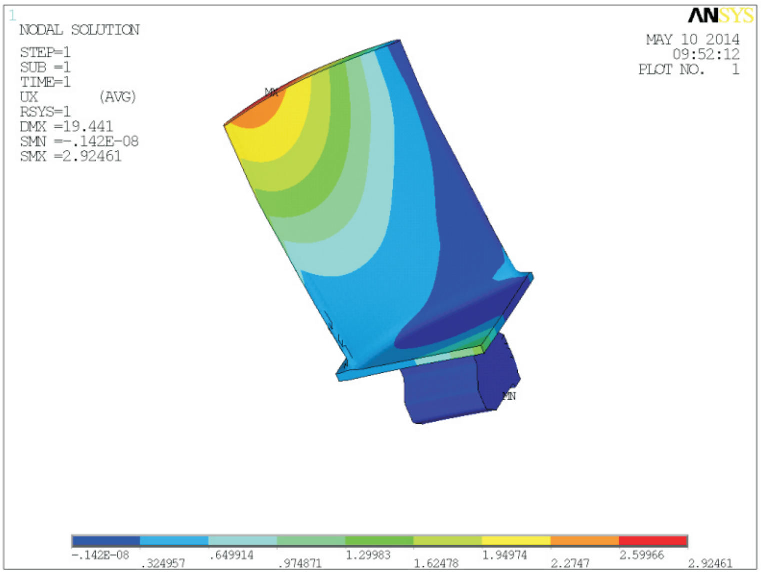

Fig 6.3: Static analysis-Displacement along X-axis

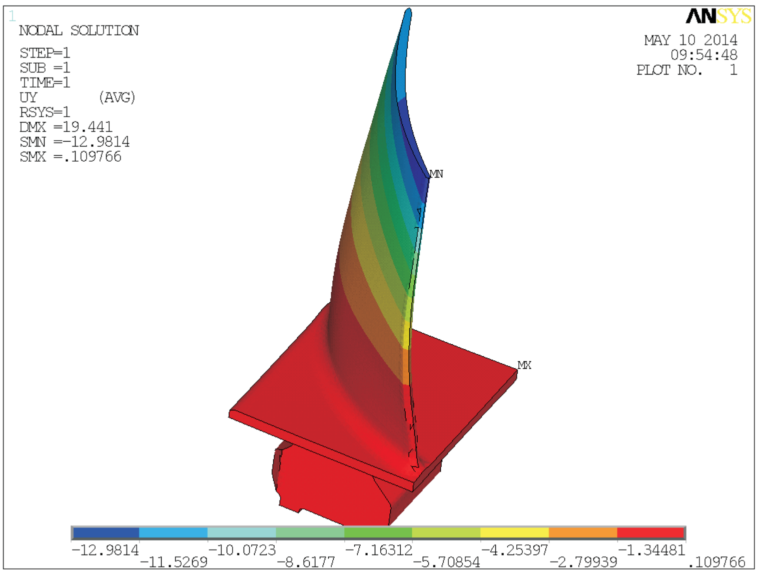

Fig 6.4: Static analysis-Displacement along y-axis

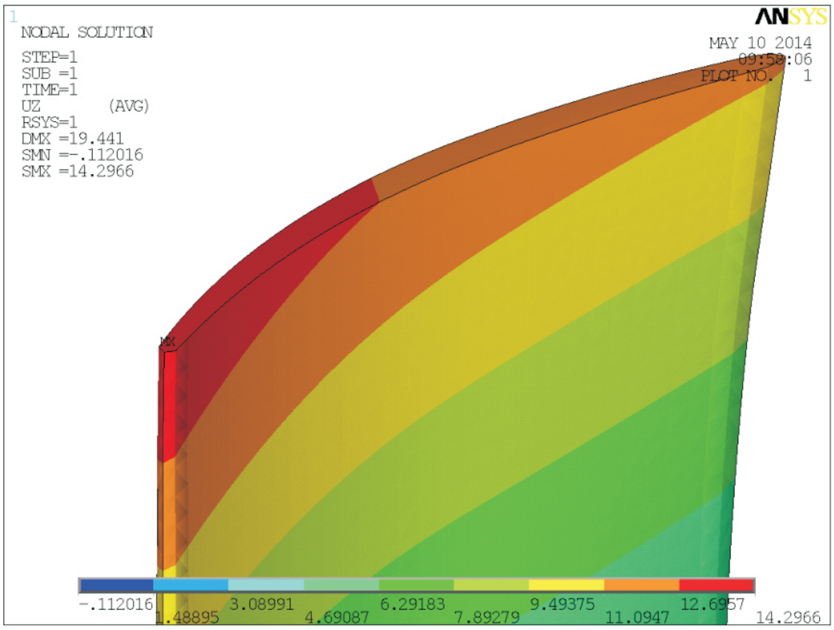

Fig 6.5: Static analysis-Displacement along z-axis

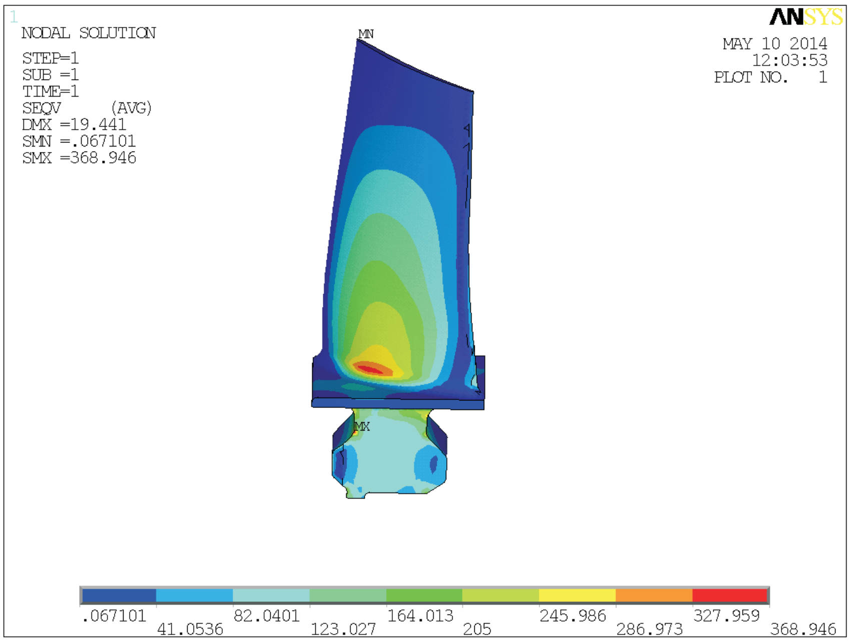

Fig 6.6: Von misses stress

6.3.2 Modal Analysis

Modal analysis is defined as the study of the dynamic characteristics of a mechanical structure. The dynamic behavior of mechanical structures is typically done using a linear system modelling approach. Modal analysis is the process of determining all the modal parameters, which are then sufficient for formulating a mathematical dynamic model.

Material of the blade: Ti-6Al-4V
Density: 4.6g/cc
Young's modulus: 1.01*10e3mpa
Poisson's ratio: 0.3

Speed of the blade: 16000rpm

In modal analysis the blade is not constrained to any of the degree of freedom; that is, the analysis is made to the blade as it is in the space and blade is made to run for the displacement and von misses stress by applying loads and the results are noted down. If the values in column time/frequency shows zeros or nearer value to the 1st six values, then the result is correct. The below figure shows the results of modal analysis.

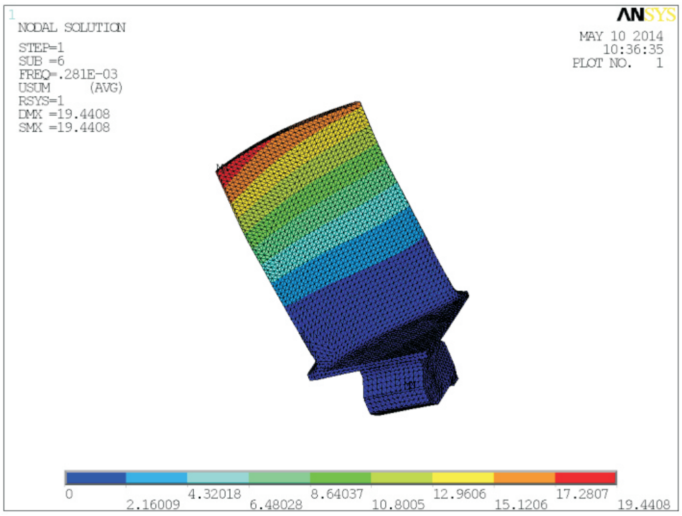

Fig 6.7: Modal analysis-max displacement

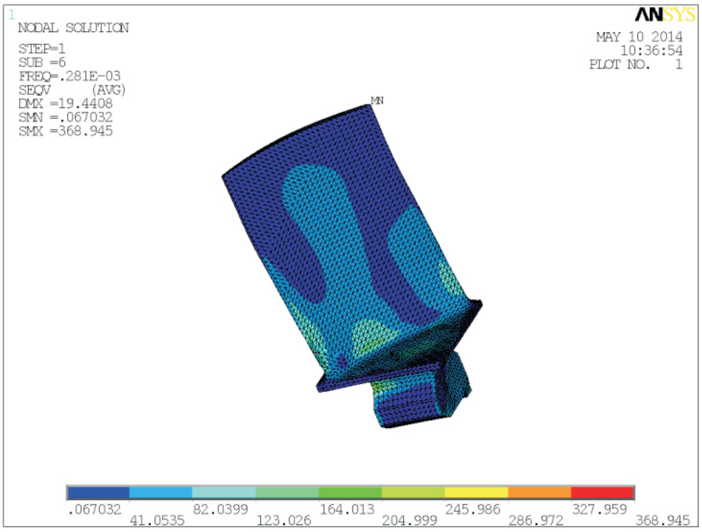

Fig 6.8: Von misses stress

7. RESULTS

7.1. Static analysis

Along X-axis: U max=2.92mm
U min=0.14e-98mm
Along X-axis: U max=0.1097mm
U min=12.98mm
Along Z- axis: U max=14.29mm
U min=0.112mm

7.2. Modal analysis

MODEL FREQUENCES				
Set	Time/ Frequency	Load Step	Sub Step	Cumulative
1	0	1	1	1
2	0	1	2	2
3	0	1	3	3
4	0	1	4	4
5	2.17E04	1	5	5
6	2.18E04	1	6	6

8. CONCLUSION

Static and Dynamic(modal) analysis are carried out
The max displacement for all the cases is reported
The model analysis is carried out and the 1st six modes are rigid body motions
The max stresses are compared with the yield strength of the material and the stresses are below the yield strength.

9. REFERENCES

1.Aero0015-1 - mechanical design of turbomachinery - 5 ects - j.-c.golinval – University of Liege (Belgium)
2.Skubbaachevskiy, G.S. Aviation gas turbine engines, construction and design of parts. Foreign Technology Edition. 1969.
3.Wilson, E.L., "Structural Analysis of Axisymmetric Solids", AIAA Journal, Volume 3, No.12, December 1965.
4.Roark, R.J., "Formulas For Stress and Strain", McGraw-Hill, 1965.
5.E. R. C. Eckert and R. M. Drake, Analysis of Heat and Mass Transfer, McGraw-Hill, New York (1972).
6.F. P. Incropera and D. P. DeWitt, Fundamentals of Heat and Mass Transfer, 2nd ed. J. Wiley & Sons, New York (1985).
7. A. H. Shapiro, The Dynamics and Thermodynamics of Compressible Fluid Flow, Ronald Press, New York (1954).

Shruthi J H
M.Tech Student, Product Design and Manufacturing, Dept IEM, Sri Siddhartha Institute of Technology, Tumkur, , Karnataka.