MODELLING, STRUCTURAL AND COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF THE BLADE OF A GAS TURBINE

[1]K. Chandra sekhar, [1]Ch.Venkata rao, [1].Sarath Kumar, [2]A.Priyanka, [1]Sathish Kumar

[1]Department of Mechanical Engineering, QIS College of Engineering and Technology, Ongole.
[2]Department of Civil Engineering, QIS College of Engineering and Technology, Ongole.

Corresponding Email: qispublications@qiscet.edu.in

Abstract

Gas turbine is one of most important turbo machines in the industrial scenario. It has a greater scope in power generation, aviation, refineries and small industries. The main theme of the paper to design, drafting a turbine blade. The using 3D softwaresolid works by using the Coordinate measuring machine point data available. The two turbine blade models are designed namely turbine blade with hole and turbine blade without hole. In order to determine the deformation and maximum shear stress values for the blades, static analysis is carried out using fluent software. Then modal analysis of the blade is carried out in order to find the mode shapes and natural frequencies. The results obtained from this work will a very useful in the field of gas turbine design.

Key words: Gas turbine blade, static Analysis, modal Analysis, Coordinate measuring machine

1. INTRODUCTION

The gas turbine is a heat engine in which air taken from the atmosphere and fuel is combusted in order to increase the gas temperature.

To study the behaviour of gas turbine blade under different loading conditions, a numerical analysis was conducted by Baqer et al. They considered different loading for the turbine blade such as 5000,10000 and 15000 N.Aluminum Alloy and Titanium alloys are considered by them.They used solidworks 18 and Ansys15 for their modelling and Analysis respectively.

Dhamecha et al and Garre et al discussed the internal cooling of the turbine blade.Prinyanka Singh et al discussed about turbine having 5,9,13,14 holes respectively on the turbine blade.

Similarly Brahmaiah et al and Harsha et al conducted research on turbine blades with holes but Brahmiah et al restricted to 13 holes and Harsha et al restricted to 12 holes.

2. METHODOLOGY

Three dimensional modelling, structural and computational fluid dynamics of gas turbine blade with holes and without holes are drafted using Catia software. Ansys Fluent is used for Computational fluid dynamics analysis (CFD). The CFD analysis is done in both turbine blade with holes and without holes.

2.1 MODELLING OF TURBINE BLADE

The three dimensioning modelling of gas turbine is done using solid works sofware. The creation and modelling of turbine blades are shown in figures from Figure 1 to Figure 8.

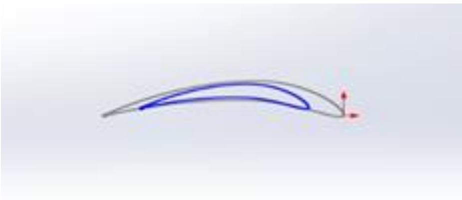

Figure1: Two airfoil profiles of turbine blade

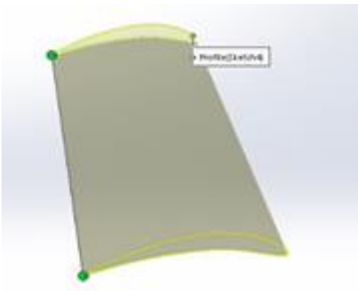

Figure2: Loft operation on turbine blade.

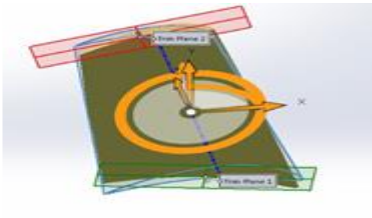

Figure3: Twist operation for turbine blade

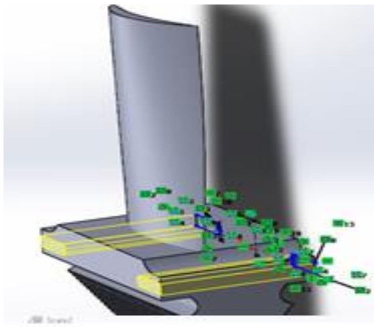

Figure 4: Creating base for turbine blade

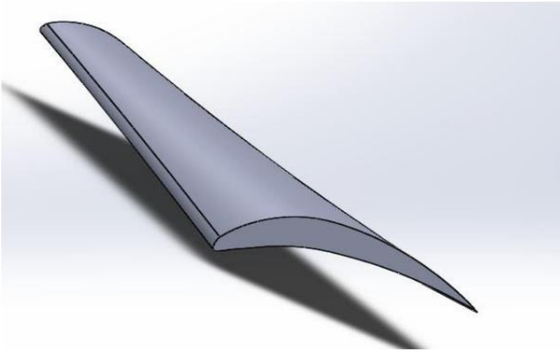

Figure 5: Turbine blade without holes

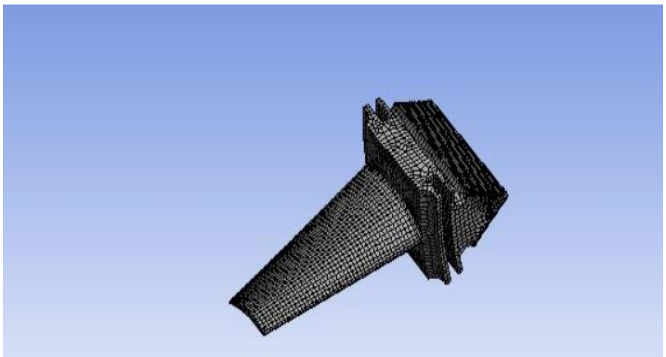

Figure 7: Model with fine meshing

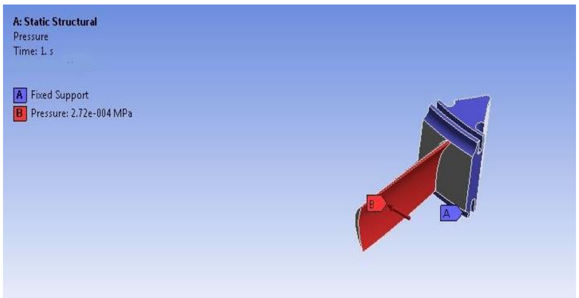

Figure 8.Turbine blade with base

2.2.1 STRUCTURAL ANALYSIS TURBINE BLADE OF GAS TIRBINE WITHOUT HOLES

The gas turbine gets its power by using the heat and pressure from burnt gases that are at high temperatures. They were expanded through the several rings of stationary and moving blades to provide power. The compressor is connected to the turbine shaft since it is powered by the turbine. The working fluid was compressed first, then expanded in a turbine. The power generated by the turbine can then be increased by increasing the volume of working fluid at constant pressure or alternatively by increasing the pressure at constant volume on the presumption that there were no losses in either component. Alternately, heat can be added to raise the temperature of the working fluid following compression. to ascend higher

The structural analysis of the turbine blade of the gas turbine of the without holes is done using Ansys Fluent and are shown in Figures from figure 9 to figure 12.

Figure 9 Total deformation of turbine blade without holes

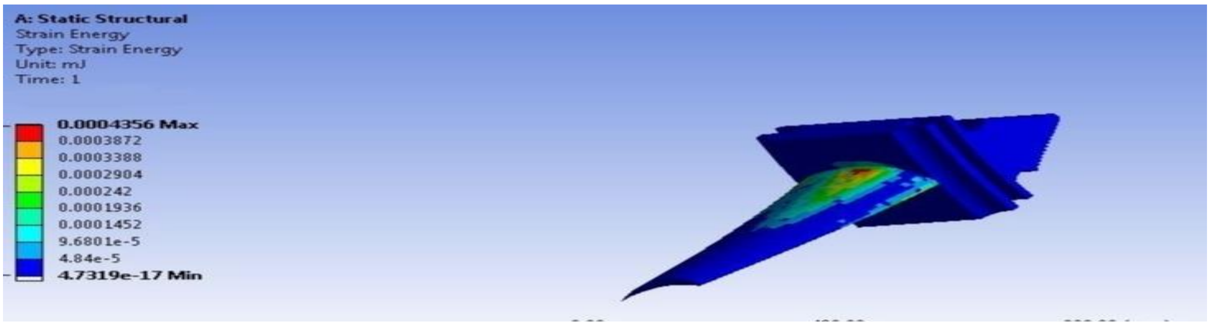

Figure10 Strain Energy of turbine blade without holes

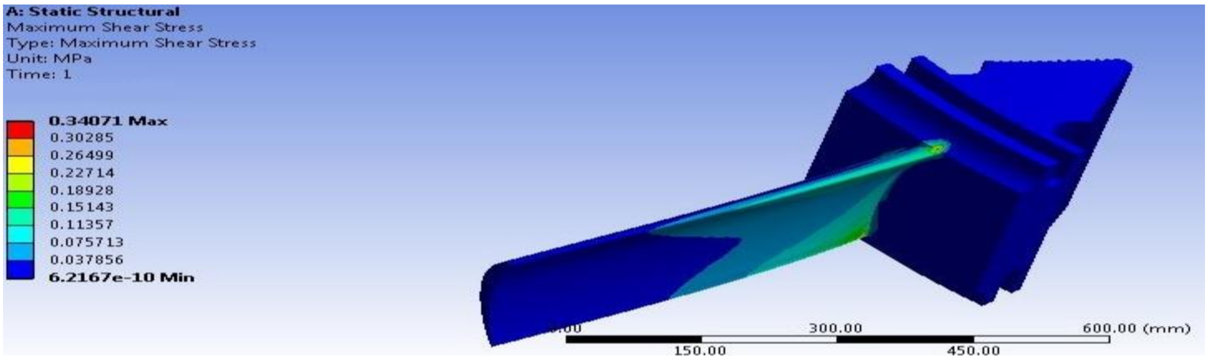

Figure11. Maximum shear stress of turbine blade without holes

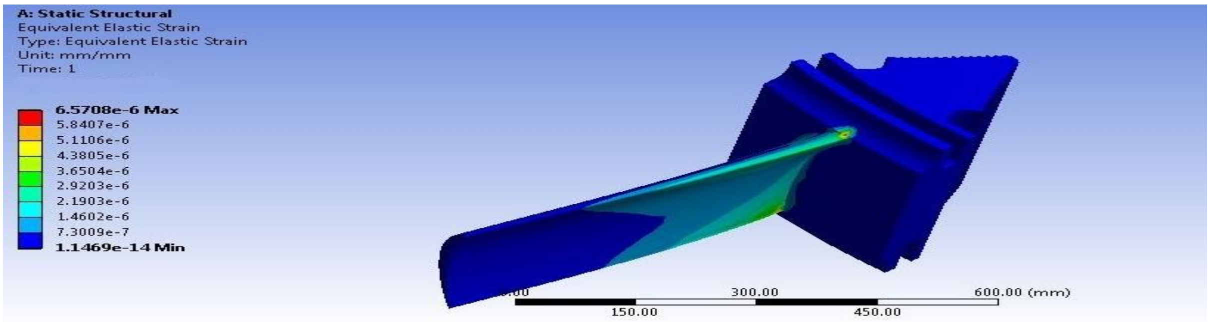

Figure12. Equivalent Elastic strain of turbine blade without holes

2.2.2 STRUCTURAL ANALYSIS OF TURBINE BLADE WITH HOLES

The turbine uses the exhaust gas to release energy. Gas turbine blade structural analysis is the focus of the work. Analysis was done to comprehend the mechanical strains and bending. This includes variables like the gas forces, which are presumed to be spread evenly, and was felt by the gas turbine rotor blade. The centroid of the blade is the point through which the tangential, axial, and radial forces act, as well as the centrifugal force, act.

The structural analysis of the turbine blade holes is carried out using Ansys Fluent and are shown in Figures from figure 13 to figure 16.

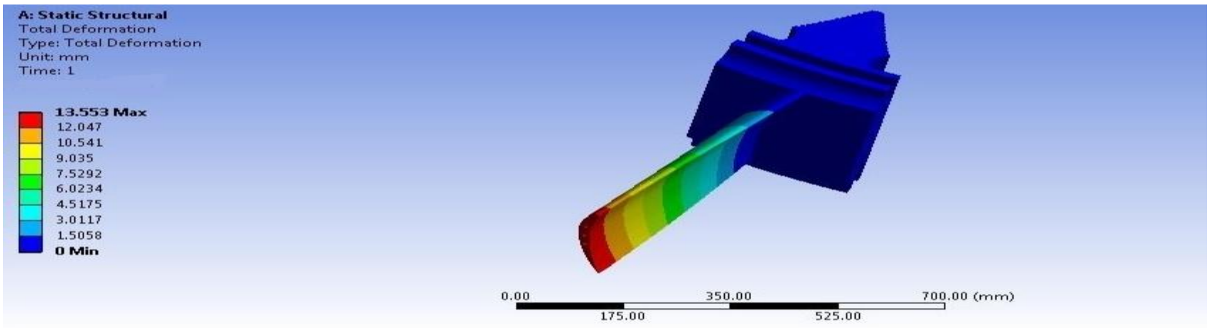

***Figure13** Total deformation of turbine blade with holes*

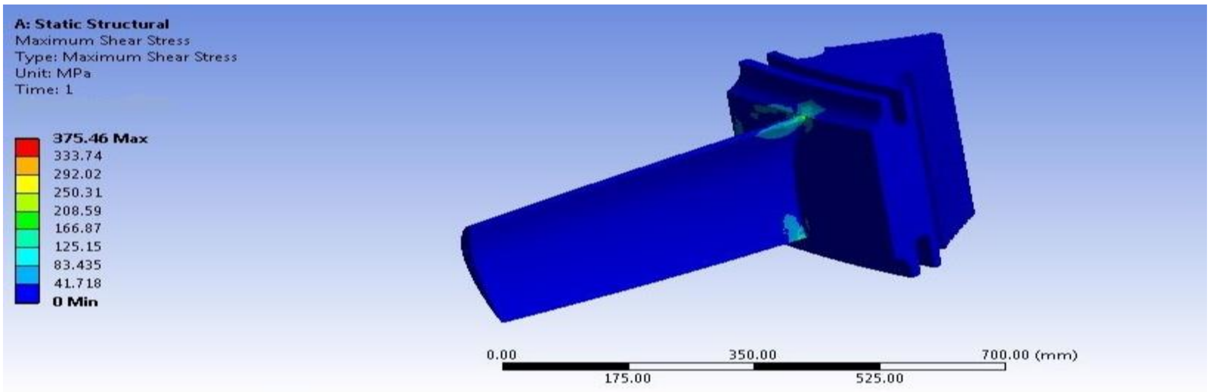

Figure 14: *Maximum Shear stress of turbine blade with holes*

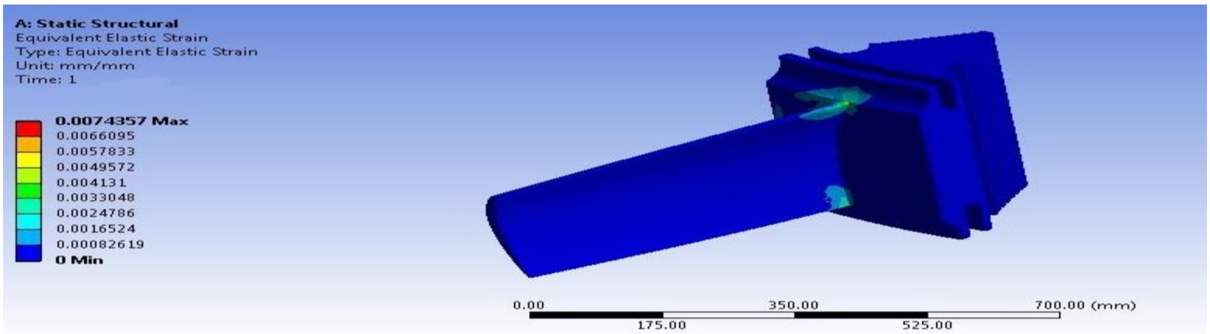

Figure15: *Equivalent elastic strain of turbine blade with holes*

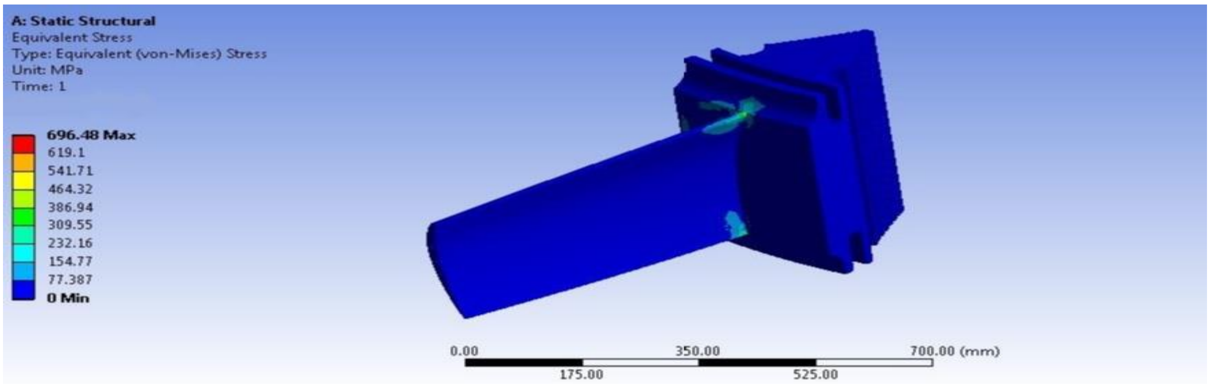

Figure16: *Equivalent stress of turbine blade with holes*

2.3.1 CFD ANALYSIS POST PROCESSING OF TURBINE BLADE WITHOUT HOLES

The collection of values for each field variable (u, v, w, p,...) at each node is the solver's raw output. It's necessary to extract and manipulate the relevant data. For instance, to calculate a drag coefficient, a subset of surface pressures, shear stresses, and cell-face areas are needed. Grid generation and flow visualisation tools are typically included by commercial CFD suppliers to their flow solvers. These tools are all controlled through a graphical user interface (GUI), which makes it easier to set up a CFD simulation and extract and manipulate data later on. The CFD analysis post processing of the turbine blade without holes is carried out using Ansys Fluent and are shown in Figures from figure 17 to figure 20.

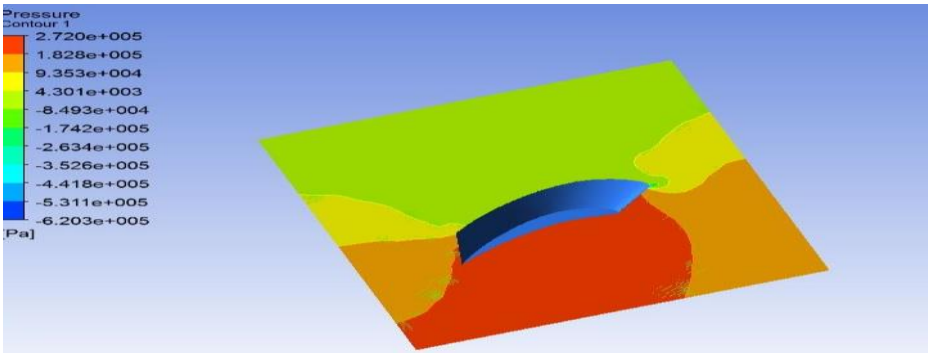

Figure17: *Pressure contour on turbine blade without holes*

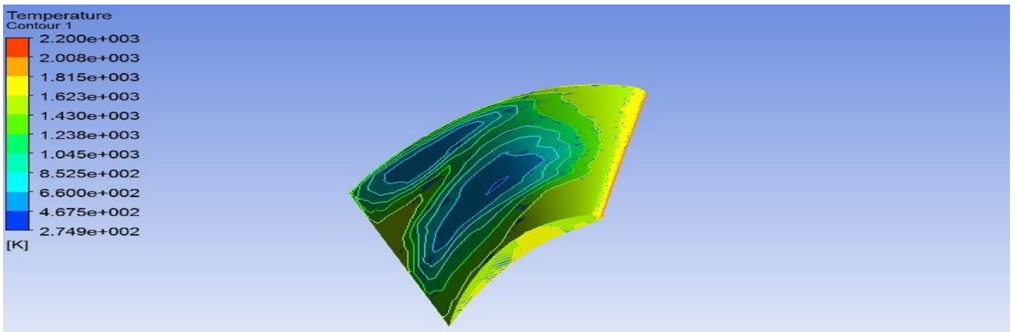

Figure 18: *Temperature contour on turbine blade without holes*

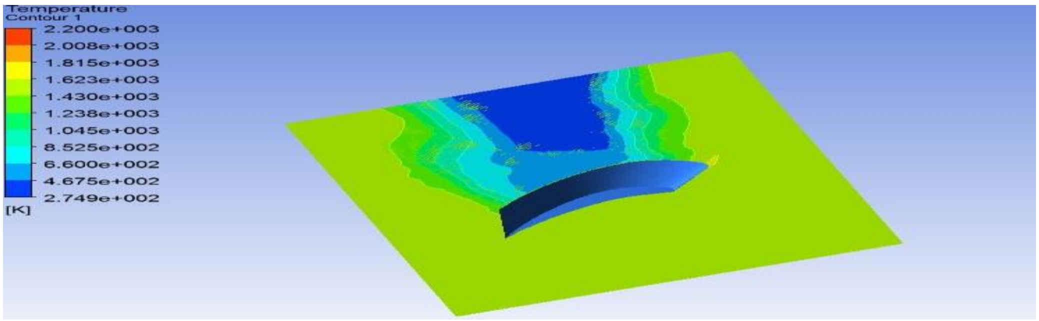

Figure 19: Temperature contour at plane o9999n turbine blade without holes

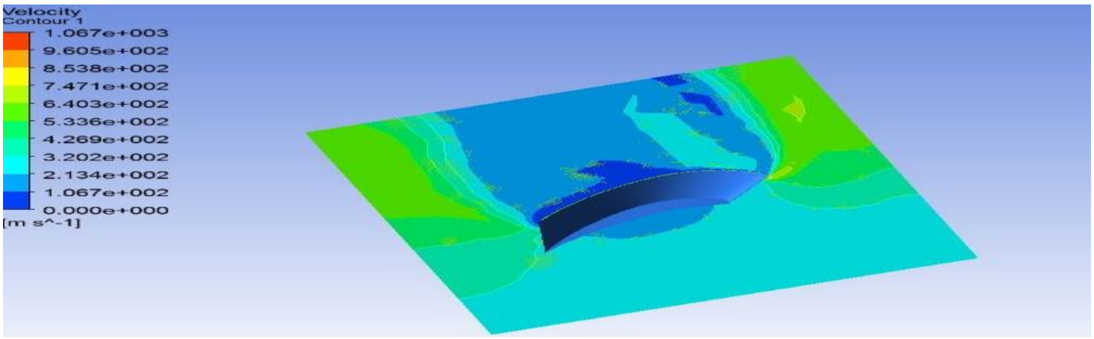

Figure 20: Velocity contour at plane on turbine blade without holes

2.3.2 CFD ANALYSIS POST PROCESSING OF TURBINE BLADE WITH HOLES

Because the built-in features are designed for engineers, architects cannot easily explore results in real time. We suggest game engine-based visualisations to fill the gap between architects and engineers. The conversion of CFD to Unity3D and the visualisation of weather data are demonstrated in this study. The CFD analysis post processing of the turbine blade with holes is carried out using Ansys Fluent and are shown in Figures from figure 21 to figure 23.

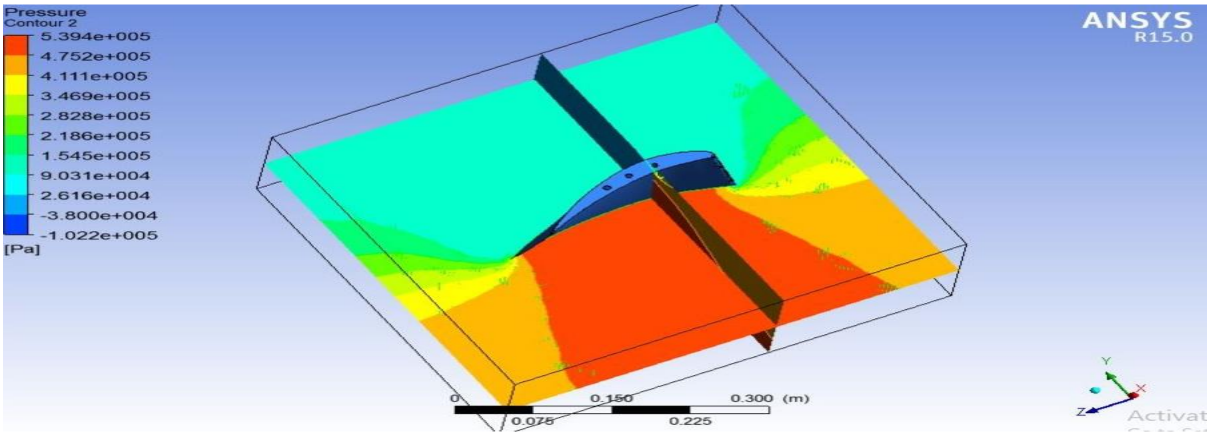

***Figure21:** Pressure Contour of turbine blade with holes*

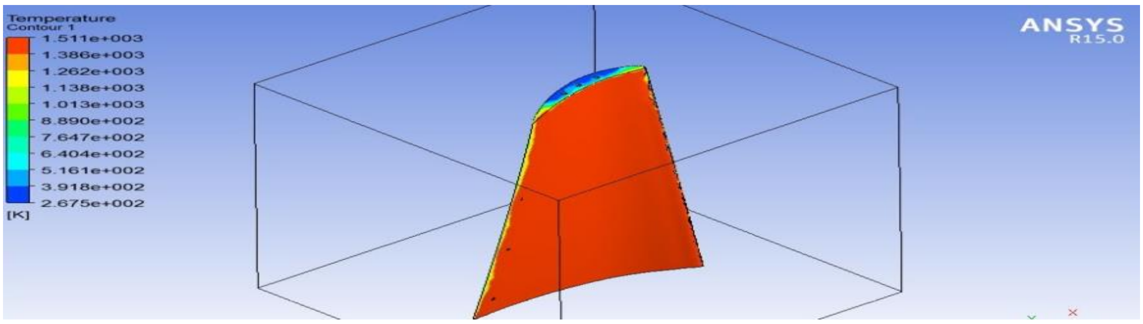

***Figure22**: Temperature Contour of turbine blade with holes*

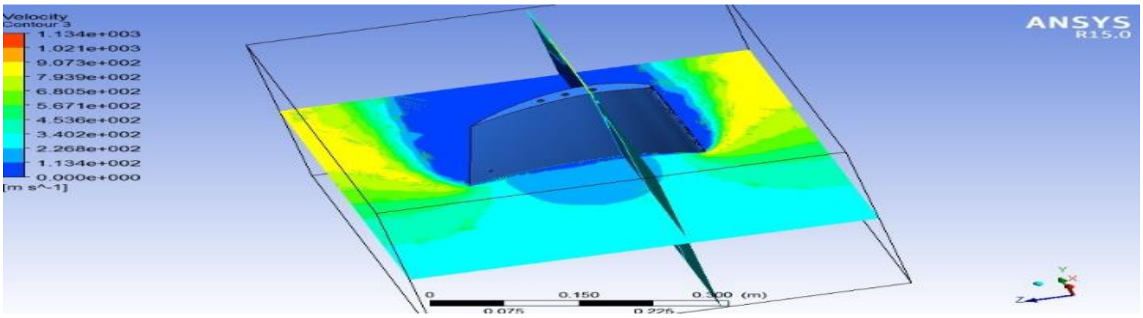

***Figure23**: Velocity Contour of turbine blade with holes*

3.1 STATIC STRUCTURAL ANALYSIS RESULTS:

TURBINE BLADE	DEFORMATION(MM)	STRESS(MPA)
WITH OUT HOLES	0.16	59
WITH HOLES	13.53	375.46

3.2 CFD RESULTS:

BLADE TYPE	PRESSURE (PA)	VELOCITY(M/S)	TEMPERATURE(K)
WITH OUT HOLES	2.7E5	106	2200
WITH HOLES	5.3E5	113	1511

4 CONCLUSION

The design of the turbine blade with blade and without blade was done in solid works software. Coordinates points of the blade profile generated the geometry is utilized for the design of the turbine blade

From the CFD results the blade with holes have less temperature profile and high velocity profile throughout the blade, Compare to the blade without holes.

In the structural analysis we are getting high deformation and stress values in the blade with holes compare to blade without holes but those values are under the factor of safety values.

From the overall comparisons we can conclude that turbine blade with holes have high performance.

5 REFERENCES

1. Nawfel M. Baqer, Hayder H. Khaleel & Noor H. Dhaher, Numerical study and analysis of gas turbine blades, International Journal of Mechanical and Production Engineering Research and Development, Vol. 9, Issue 3, Jun 2019, 959-966
2. Lalit Dhamecha , Shubham Gharde , Ganraj More , M.J.Naidu, Design and Analysis of Gas Turbine Blade with Varying Pitch of Cooling Holes, International Journal of Engineering Trends and Technology (IJETT) – Volume 34 Number 8- April 2016.
3. Parthasarathy Garre* and Chikateri Susmitha[1],Numerical simulation of gas turbine blade cooling, IOP Conf. Series: Materials Science and Engineering 455 (2018) 012012
4. Priyanka Singh&O P Shukla, Heat Transfer Analysis of Gas Turbine Rotor Blade Cooling Through Staggered Holes using CFD, International Journal of Engineering Research & Technology (IJERT) Vol. 4 Issue 07, July-2015
5. K HariBrahmaiah, M.LavaKumar2, Heat Transfer Analysis of Gas Turbine Blade Through Cooling Holes, International Journal of Computational Engineering Research, International Journal of Computational Engineering Research, Vol-04 || Issue, 7 || July – 2014
6. Harsha D A, Yogananda A, CFD analysis on radial cooling of gas turbine blade, international journal of innovations in engineering research and technology, volume 4, issue 6, june-2017