International Journal on Mechanical and Production Engineering
Volume 4: Issue 2 : April 2018, pp 1 – 7 www.aetsjournal.com *ISSN (Online) : 2395 - 3500*

THERMAL RESISTANCE FOR IMPROVE THERMAL EFFICIENCY IN GAS TURBINE BLADES

V.SATHISHKUMAR , B.PANDIYARAJAN , S.MATHIYAZHAGAN
N.MATHIAZHAGAN

Abstract**—** 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 Pro-E wildfire is used for deign of solid model of the turbine blade with the help of the spline and extrude options ANSYS software is used analysis of F.E. model generated by meshing of the blade using the solid brick element present in the ANSYS software itself and thereby applying the boundary condition.In this project, Design analysis of thermal resistance nano coating for improved thermal efficiency in gas turbine blades. Due to various losses such as thermal and corrosion (hot & cold corrosion) losses affect the gas turbine so blade life is reduced. Already they had conducted many experiments such as cooling holes in blades and coating is done on the blade. But above experiments were failures.Toavoid such failures, we were using the nano coating or composite materials to the blade for increasing the blade life.

Keywords -- FEA , ANSYS , Turbine blade , Nano coating

I. INTRODUCTION

ANSYS stands for analysis system product. Dr. john Swanson founded ANSYS inc in 1970 with vision to commercialize the concept of computers simulated engineering establishing himself as one of the pioneers of finite element analysis (FEA). ANSYS Inc supports the ongoing development of innovative technology and delivers flexible, enterprise-wide engineering systems that enable companies to solve the full range of analysis problem, maximizing their existing investments in software and hardware. ANSYS continues its roles as a technological innovator. It also supports a process centric approach to design and manufacturing allowing users to avoid expensive and time consuming " build and brake" cycles ANSYS analysis and simulation tools give customers ease of use data capability, multi platform support and coupled field multi physics capabilities.Turbines in today's world of power and aviation. It has proved itself to be the numerous in spite of stiff competition. The most attractive feature however is that the performance can still be improved.

Theoretical Analysis of Gas Turbine blades by finite element method by Lawrence and Depanraj [1] published various techniques for cooling of turbine blades. 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, 12) where 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 plotted for temperature distribution for existing design (12 holes) and for 8 holes against time. a turbine measurements. Areas of large heat transfer rates are identified and physical reasoning for the phenomena presented blade with 8 holes configuration is found to be the optimum solution.

Transient state stress analysis on an axial flow gas turbine blades and disk using finite Element procedure by Sukhvinder Kaur bhatti and Niranjan Kumar [2] published the maximum stresses obtained from different analysis by using innovative high heat resistant material INCONEL 718 are found to be within the yield strength of the material. Interesting results obtained in terms of maximum operational radial stress, maximum operational hoop stress, maximum operational Von mises stress, the temperature field etc.

Kim (2008), in their analysis of failure in J85 engine turbine blades[3] stated 7 that after observing the engine exterior upon landing, cracks and deformations were found in many parts including the i inlet manifold and the afterburner fuel manifold. After removing parts and carrying out SEM analysis on cracked parts, the problem was found to be secondary cracks due to overstress.They also mention about the engine vibration and external parts damage were caused by fragments created when the seven blades suffered fractures 1/3 of the way down from the tip as identified on 2 of the first stage turbine rotor blades resulting in damages to parts of the other

V.Sathishkumar, Assistant Professor , Meenakshiramaswamy Engineering College , Thathanur , Tamilnadu, India.
B.Pandiyarajan , Assistant Professor ,Meenakshiramaswamy Engineering College, Thathanur,Tamilnadu,India.
S.Mathiyazhagan , HOD, Mechanical, Meenakshi Ramaswamy Engineering College ,Thathanur,Tamil Nadu. India.
(Email id: mathiyazhagan4690@gmail.com)
Dr.N.Mathiazhagan , Principal, Meenakshi Ramaswamy Engineering College, Thathanur ,Tamilnadu, India.
(Email id: mathi.sharmi@gmail.com)

66 blades. The first stage turbine blades, where the initial failure occurred, suffered failure because over the long operating time, the coating peeled off due to high temperature air on the concave side surface, followed by direct exposure of the base metal to high temperature resulting in creep rupture.

T.J Carter (2005) in the research about common failures in gas turbine blades [4] stand with the opinion that there are three probable damage mechanisms affect turbine blades, these being mechanical damage through either creep or fatigue and high temperature corrosion. The use of light alloys for the high temperature sections of the engine is not feasible since they cannot generally be design to give acceptable creep properties at the high temperatures needed for efficient turbine operation.

II. EXPERIMENTAL PROCEDURE

The ANSYS program is a flexible, robust design analysis and optimization package. The software operates on major computers and operating system, from pc to work stations to super computers ANSYS features file computability thought the family of the products and across all platforms.

ANSYS design data access enables user to import computer aided design models in to ANSYS, eliminating repeat work. This ensures enterprise wide, flexible engineering solution for all ANSYS user.

1) Processor

ANSYS function are organized are in to two groups called processors. The ANSYS program has one pre-processor, one solution processor,two-post processor, and several auxiliary processors such as the design optimizer.The ANSYS pre-processor allows the user to create o finite element model and to specify options needed for a subsequent solution. The solution processor is used to apply the loads and the boundary conditions, and then determine the response of the model to them. With the ANSYS post processor, the user retrieves and to perform additional calculations of interest.

2) Database

The ANSYS program uses a single, centralized database for all model data and solution results. Model data (including solid model and finite element model geometry, materials etc.) are written to the database using the processor. Loads and solution results data are written using the solution processor. Post processing results data are written using the post processors. Data written to the database while using one processor is therefore available, as necessary, in the other processors. File Format

Files are used, when necessary, to pass the data from part of the program to another, to store the program the database, and to store program output.

3) Analysis Types Available

The following types of analysis are possible using ANSYS Structural Analysis: Static analysis, Buckling analysis, Harmonic, Non linear and Transient analysis. Thermal Analysis: Steady state thermalAnalysis, Transient thermal Analysis.

CFD (Computational Fluid Dynamics) Analysis Laminar or turbulent or adiabatic, free surface, compressible or incompressible, Newtonian or Non-Newtonian, multiple species transport.

Material Definition: All elements are defined by nodes, which have only their location defined. In the case of plate and shell elements, there is no indication of thickness. The thickness can be given as element property. Properties table for a particular property set 1D have to be input. Different types of elements have different

4) Properties for example:

Beams: Cross-sectional area, moment of inertia etc;
Shells: Thickness
Springs: Stiffness

The user also needs to define material properties of the elements. For linear static analysis, modulus of elasticity and poison's ratio need to be provided. For heat transfer coefficient of thermal expansion, densities etc; are required

5) Solution

The solution phase deals with the solution of the problem according to the problem definitions. All the tedious work of formulating and assembling of matrices are done by the computer, and finally displacements and stress values are given as output. Some of the capabilities ANSYS are linear static analysis, transient dynamic analysis etc.

TABLE1.ELEMENT SHAPES

Element Order	2D Solid	3D Solid	3D Shell	Line Elements
Linear	PLANE42 PLANE182	SOLID45 SOLID185	SHELL63 SHELL181	BEAM3/44 BEAM188
Quadratic	PLANE82/183 PLANE2	SOLID95/186 SOLID92/187	SHELL93	BEAM189

6) Post processor

It is a powerful user-friendly Post processing program. Using interactive colour graphics, it has extensive plotting features for displaying results obtained from thefinite element analysis. One picture of analysis results (i.e. results in a visual form)can often reveal in seconds what would take an engineer hour to asses from a numerical output, say in a tabular form. The engineer may also see important aspects of the results that could be easily missed in a stack of numerical data

The first step in creating geometry is to build a solid model of the item you are analyzing. You can use either predefined geometric shapes such as circles and rectangles (known within

ANSYS as primitives), or you can manually define nodes and elements for your model. The 2-D primitives are called areas, and 3-D primitives are called volumes. Model dimensions are based on a global coordinate system. By default, the global coordinate system is Cartesian, with X, Y, and Z axes; 31Modeling also uses a working plane - a movable reference plane used to locate and orient modeling entities. We can turn on the working plane grid to serve as a "drawing tablet" for your model.

Applying Loads and Obtaining the Solution You must define the analysis type and options, apply loads to the model, specify load step options, and initiate the finite element solution. Defining the Analysis Type During this phase of the analysis, you must first define the analysis type In the GUI, choose menu path Main Menu Solution> Analysis Type> New Analysis> Steady state (static). If this is a new analysis, issue the command ANTYPE, STATIC, NEW.

7) Applying Loads

You can apply loads either on the solid model (key points, lines, and areas) or on the finite element model (nodes and elements). You can specify loads using the conventional method of applying a single load individually to the appropriate entity, or you can apply complex boundary conditions as tabular boundary conditions or as function boundary conditions Constant Temperatures (TEMP)

These are DOF constraints usually specified at model boundaries to impose a known, fixed temperature. Heat Flow Rate (HEAT) These are concentrated nodal loads. Use them mainly in line-element models (conducting bars, convection links, etc.) where you cannot specify convections and 32heat fluxes. A positive value of heat flow rate indicates heat flowing into the node (that is, the element gains heat). If both TEMP and HEAT are specified at a node, the temperature constraint prevails.

8) Convections (CONV)

Convections are surface loads applied on exterior surfaces of the model to account for heat lost to (or gained from) a surrounding fluid medium. They are available only for solids and shells. In line-element models, you can specify convections through the convection link element(LINK34).

9) Materials for turbine blade

The use of high gas temperature at the turbine entry is intimately linked with the materials that can be used in such applications. The following properties are required in the high temperature materials employed in gas turbines.

1. High strength at the maximum possible temperature.
2. Low creep rate
3. Resistance to corrosion and oxidation
4. Resistance to fatigue and

Steel Alloys offer a number of advantages in the manufacture of gas turbine components. They generally have high percentages of nickel and chromium and can be used upto temperatures of 650 oC.Aluminium and its various alloys with their low density and workability are ideal for castings and forgings and can be used up to 260°C.Titanium and its alloys are also light and can withstand temperatures up to 550°C.Gas turbine blades at high temperatures work in an atmosphere that is both corrosive and oxidizing. Therefore, for temperatures between 650°C and 950°C nickel and chromium based alloys are used. They have high strength and low creep combined with good ductility. The shock resistance of such alloys is also high. Cobalt alloys have high strength and resistance to oxidation up to temperatures of 1150°C.Other alloys specially developed for gas turbine blades operating at high temperatures havemanganese, molybdenum, copper, silicon, tungsten, vanadium and zirconiumHigh temperature corrosion resistant alloys: These have relatively low hot strength (for a nickel alloy) but good scaling resistance.

They include:

a) Nimonic 75

b) Inconel 600 and 601 with higher scaling resistance section of the solidifying metal in advance of the molded blade or by using a seeding crystal.

c) 50/50 nickel chrome available as castings as a cladding material (incoclad) and IN567 (with niobium) available as castings are the materials with the highest resistance to fuel ash corrosion.

Fatigue strength and oxidation resistance. The small margin between maximum preheating temperature (1100°C) and minimum working temperature (1050°C) of Nimonic 115 indicates that this is likely to remain the forged alloy with the best high temperature properties. Vacuum casting provides an alternative manufacturing capability required the addition of a high melting point and solid solution strengtheners such as tungsten. This and the alloy IN138 which, although it lacks the best creep rupture properties, is extremely well suited to coating with aluminum and is therefore used where the maximum creep resistance is not essential, where standardized. The next improvement in temperature capability came with directionally solidified blades.

10) Widely used alloys

The most widely used alloys for turbine blades, their developer or user and their composition is shown.

TABLE2. WIDELY USED ALLOYS COMPOSITION

ALLOY	Co	Cr	Al	Ti	C	Mo	W	Fe	Zr
INCONEL 722	-	15	0.6	2.5	0.04	-	-	7	-
NIMONIC 115	14.8	15	5	4	0.25	4	-	1	0.2
IN738	8.5	16	3.4	3.4	0.17	1.8	2.6	0.5	0.1
RENE 80	9.5	14	3	5	0.17	4	4	-	0.03
B1900	10	8	6	1	0.025	6	-	-	0.04

TABLE 3. MATERIALS SELECTED FOR ANALYSIS AND THEIR PROPERTIES

Property	Unit	N-155	Mild Steel	INCONEL 718
E	Pa	143E09	2.1E11	149E09
P	Kg/m3	8249	7860	8220
K	W/m-K	20	28	25
M	--	0.344	0.29	0.331
A	E-06/°C	17.7	6.12	16
Yield stress	MPa	550	415	1067

Steel is any alloy of iron, consisting of 0.2% to 2.1% of carbon, as a hardening agent. Besides carbon, many other metals are a part of it.

TABLE 4. CHEMICAL COMPOSITION OF N155

Element	Min	Max
Carbon	0.08	0.16
Manganese	1	2
Silicon	--	1
Sulfur	--	0.03
Chromium	20	22.5
Nickel	19	21
Copper	--	0.5
Tungsten	2	3
Iron	Balance	
Cobalt	18.5	21

11) Inconel 718

Alloy 718 is a precipitation hardenable nickel-based alloy designed to display exceptionally high yield, tensile and creep-rupture properties at temperatures up to 1300°F.

This alloy has been used for jet engine and high-speed airframe parts such as wheels, buckets, spacers, and high temperature bolts and fasteners°C.

III. RESULT AND DISCUSSION

A thermal analysis calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities of interest are

1.The temperature distributions
2.The amount of heat lost or gained
3.Thermal gradients
4.Thermal fluxes.

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components.

Only the ANSYS Multiphysics, ANSYS Mechanical, ANSYS Professional, and ANSYS FLOTRAN programs support thermal analyses. The basis for thermal analysis in ANSYS is a heat balance equation obtained from the principle of conservation of energy. The finite element solution you perform via ANSYS calculates nodal Temperatures, then uses the nodal temperatures to obtain other thermal quantities. The ANSYS program handles all three primary modes of heat transfer: conduction, convection, and radiation.

1) CONVECTION

You specify convection as a surface load on conducting solid elements or shell elements. You specify the convection film coefficient and the bulk fluid temperature at a surface; ANSYS then calculates the appropriate heat transfer across that surface. If 21 the film coefficient depends upon temperature, you specify a table of temperatures along with the corresponding values of film coefficient at each temperature. For use in finite element models with conducting bar elements (which do not allow a convection surface load), or in cases where the bulk fluid temperature is not known in advance, ANSYS offers a convection element named LINK34.

2) RADIATION

ANSYS can solve radiation problems, which are nonlinear, in four ways

1. By using the radiation link element, LINK31
2. By using surface effect elements with the radiation option (SURF151in 2-D modeling or SURF 152in 3-D modeling)
3. By generating a radiation matrix in AUX12 and using it as a super element in a thermal analysis.
4. By using the Radiosity Solver method.

3) TYPES OF THERMAL ANALYSIS

ANSYS supports two types of thermal analysis A steady-state thermal analysis determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored. A transient thermal analysis determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

Coupled-Field Analysis Some types of coupled-field analyses, such as thermal-structural and magnetic-22 thermal analyses, can represent thermal effects coupled with other phenomena. A coupled-field analysis can use matrix coupled ANSYS elements, or sequential load-vector coupling between separate simulations of each phenomenon. Steady-State Analysis The ANSYS Multiphysics, ANSYS Mechanical, ANSYS FLOTRAN, and ANSYS Professional products support steady-state thermal analysis. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineer/analysts often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. We can use steady-state thermal analysis to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convections
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material do vary with temperature, so the analysis usually is nonlinear. Tasks in a Thermal Analysis The procedure for doing a thermal analysis involves three main tasks: Build the model. Apply loads and obtain the solution. Review the resultsBuilding the Model To build the model, you specify the job name and a title for your analysis. Then, you use the ANSYS preprocessor (PREP7) to define the element types, element real constants, material properties, and the model geometry.

For a thermal analysis, you also need to keep these points in mind:

1. To specify element types, you use either of the following:
2. To define constant material properties, use either of the following:

To define temperature-dependent properties, you first need to define a table of temperatures. Then, define corresponding material property values. Creating Model Geometry

There is no single procedure for building model geometry; the tasks you must perform to create it vary greatly, depending on the size and shape of the structure to model. The first step in creating geometry is to build a solid model of the item you are analyzing. You can use either predefined geometric shapes such as circles and rectangles (known within ANSYS as primitives), or you can manually define nodes and elements for your model. The 2-D primitives are called areas, and 3-D primitives are called volumes.

Model dimensions are based on a global coordinate system. By default, the global coordinate system is Cartesian, with X, Y, and Z axes Modeling also uses a working plane - a movable reference plane used to locate and orient modeling entities. We can turn on the working plane grid to serve as a "drawing tablet" for your model.

Defining the Analysis Type During this phase of the analysis, you must first define the analysis type:

1. In the GUI, choose menu path Main Menu Solution> Analysis Type> New Analysis>
2. If this is a new analysis, issue the command ANTYPE, STATIC, NEW.

Applying Loads You can apply loads either on the solid model (key points, lines, and areas) or on the finite element model (nodes and elements). You can specify loads using the conventional method of applying a single load individually to the appropriate entity, or you can apply complex boundary conditions as tabular boundary conditions or as function boundary conditions. Thermal load can be of any of the following type.

Constant Temperatures (TEMP) These are DOF constraints usually specified at model boundaries to impose a known, fixed temperature. Heat Flow Rate (HEAT) These are concentrated nodal loads. Use them mainly in line-element models (conducting bars, convection links, etc.) where you cannot specify convections and heat fluxes. A positive value of heat flow rate indicates heat flowing into the node (that is, the element gains heat). If both TEMP and HEAT are specified at a node, the temperature constraint prevails.The geometric modeling of the turbine was done using the modeling tool PRO-E. The simulation of the turbine blade behaviour is carried out using finite element software ANSYS for simulating the blades .The analysis of structural and thermal stress values that are produced while the turbine is operating are the key factors of study while designing the next generation gas turbines. The present study examines structural, thermal analysis of the first stage rotor blade of gas turbine.

The design features of the turbine segment of the gas turbine have been taken from the preliminary design of a power turbine, since the allowable temperature on the turbine blade dependents on the hot gas temperatures from the combustion chamber. The 3-D Finite Element models are developed with governing boundary conditions and solved. As the temperature has a significant effect on the overall stress on the rotor blades, a detail study on mechanical and thermal stresses are estimated and evaluated with the experimental values. Since the turbine blade is fixed to the rotor, the nodes at the lower end of the blade are constrained in all DOF. With these boundary and loading conditions, the displacements and stresses of blade are found out. The blades are heated by primary flow of gases .The temperature of the combustion gas at nozzle entry is fixed at 1031°C.Since the turbine used here is impulse type , most of the temperature drop takes place within the nozzle is around 916.644°C. Then nano coated gas turbine blade nozzle entry is fixed at 1031°C.temperature drop takes place within the nozzle is around 721.13°C.

4) MODELING DIAGRAM IN PRO-E

International Journal on Mechanical and Production Engineering
Volume 4: Issue 2 : April 2018, pp 1 – 7 www.aetsjournal.com *ISSN (Online) : 2395 - 3500*

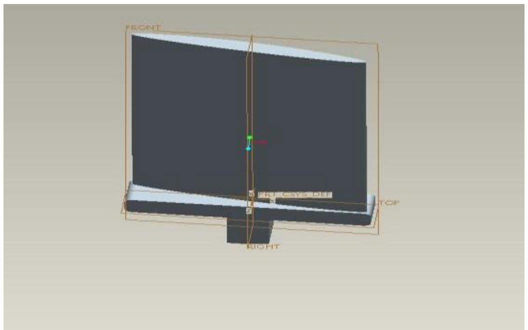

Figure 1. Modeling diagram in pro-e

5) GEOMETRY MODEL

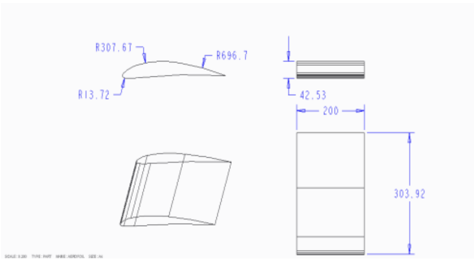

Figure 2.Geometry Model

There is no single procedure for building model geometry; the tasks you must perform to create it vary greatly, depending on the size and shape of the structure to model.

6) THERMAL ANALYSIS

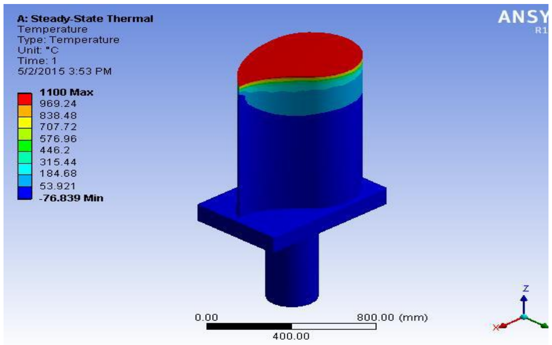

Figure 3.Thermal analysis

The Analysis Result (T)= 969.24°C.

7) THERMAL ANALYSIS (WITH NANO COATING)

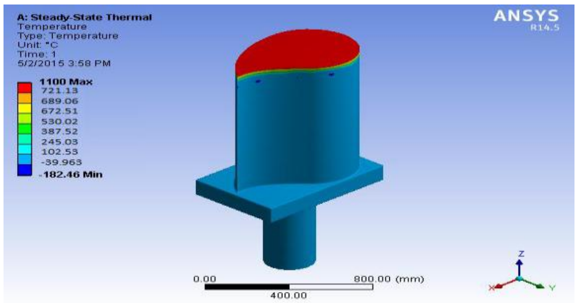

Figure 4.Thermal analysis (with nano coating)

The Analysis Result (T)= 721.13°C

8) COMPOSITE MATERIALS -THERMAL ANALYSIS IN N- 66

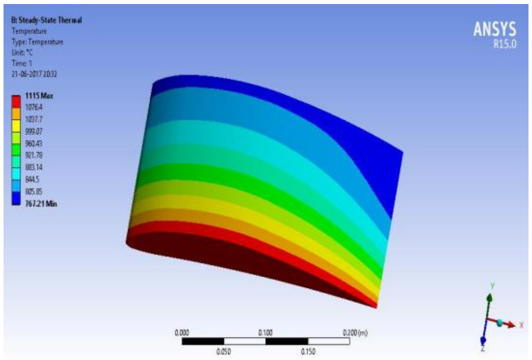

Figure 5.Thermal analysis in n- 66

The Analysis Result (T)= 767.21°C.

9) DEFORMATION ANALYSIS IN N- 66

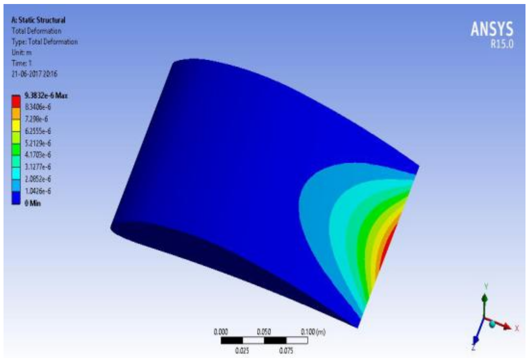

Figure 6. Deformation analysis in n- 66

The Analysis Result (Deformation)= 9.3832 e-6 m.

N-66 is a polyamide made by polycondensation of adipic acid methylenediamine and contains a total of 12 carbon atoms in each repeating unit.

10) THERMAL ANALYSIS IN KELVAR

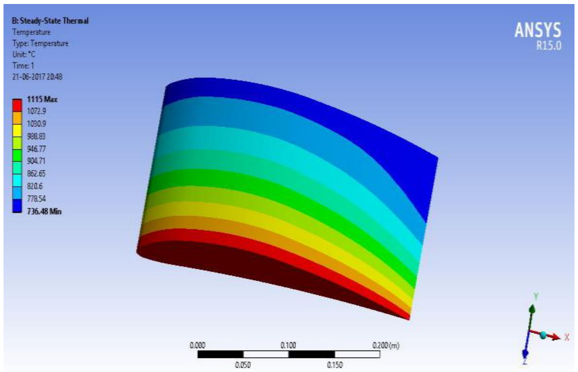

Figure 7. Thermal analysis in kelvar

The Analysis Result (T)= 736.48°C.

11) DEFORMATION ANALYSIS IN KELVAR

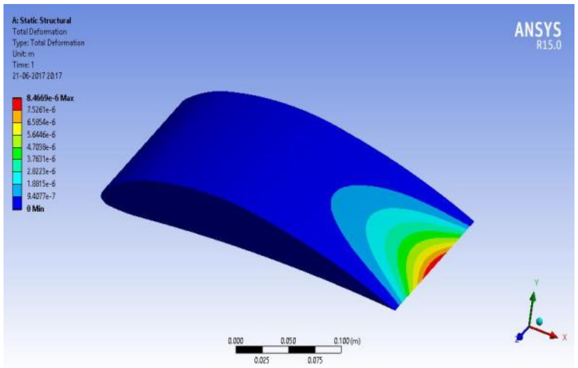

Figure 8.Deformation analysis in kelvar

The Analysis Result (Deformation)= 8.4669 e-6 m.

IV. CONCLUSION

A steady state gas flow analysis was carried out by means of ANSYS software. Then, by mapping these results, the stress analysis was carried out. Temperature and stress contours and the magnitude and displacement showed consistency with real conditions.

In this project we have analyzed previous designs and generals of turbine blade to do further optimization, Finite element results for free standing blades give a complete picture of structural characteristics, which can be utilized for the improvement in the design and optimization of the operating conditions.

REFERENCES

[1] A.K.Mattaet.al. studied the stress analysis for N – 155 & Inconel 718 material.
[2] B.Deepanraj,P. Larence and G.Sankaranarayanan. “Theoretical Analysis of Gas Turbine Blade by Finite Element Method”, *Scientific world*.vol.9.no.9, July 2011.
[3] Dr. K. Ramachandra**,** Director, Gas Turbine Research Establishment, Bangalore describes need for analysis of stress concentration factor.
[4] G.Narendranath and S.Suresh“Thermal Analysis Of A Gas Turbine Rotor Blade By Using Ansys”, International Journal of Engineering Research and Applications, Vol. 2, Issue 5, September- October 2012, pp.2021-2027.
[5] Kim et al. (2008), analysis of failure in J85 engine turbine blades.
[6] Niranjankumar. “Transient State Stress Analysis on An Axial Flow Gas Turbine Blade and Disk Using Finite Element Procedure.
[7] R D V Prasad, G Narasa Raju Steady State Thermal & Structural Analysis of Gas Turbine Blade Cooling System.
[8] Sukhvinderkaurbhatti, Shyamala kumara, M L Neelapu, C.kedarinath,
[9] T.J Carter (2005) in the research about common failures in gas turbine blades.