CFD Analysis of a Gas Turbine Blade Cooling in the Presence of Holes

Aqeel Jomma Athab
ME Student,
Department of Mechanical Engineering,
JNTUH College of Engineering,
Hyderabad-India.

Dr.S.Naga Sarada
Professor,
Department of Mechanical Engineering,
JNTUH College of Engineering,
Hyderabad-India.

ABSTRACT:

Gas turbines are extensively used for air craft propulsion, land based power generation and industrial applications. Thermal efficiency of gas turbine improved by increasing turbine rotor inlet temperature. The current rotor inlet temperature in advanced gas turbine is for above the melting point of blade material. A sophisticated cooling scheme must be developed for continuous safe operation of gas turbines with high performance. Gas turbines are cooled externally and internally. Several methods have been suggested for the cooling of blades and vanes. The techniques that involve to cool the blades and vanes by using cooling methods is to have radial holes to pass high velocity cooling air along the blade span. In this thesis, a turbine blade is designed and modeled in CATIA v5 software. The turbine blades are designed using cooling holes. The turbine blade is designed with no holes, 4 holes, 8 holes and 12 holes. CFD analysis is done to determine the pressure distribution, velocity, temperature distribution and heat transfer rate by applying the inlet velocities. Thermal and Structural analysis is done to determine the heat transfer rates and strength of the blade. The present used material for blade is chromium steel. In this thesis, it is replaced with Inconel 718 and N-155. The better material for blade is analyzed.

Keywords:

CFD analysis, Thermal and Structural analysis is done in Ansys 14.5.

1.Introduction:

With the advent in Gas turbine technology, its usage as a prime mover has become prominent, since last few decades. One of the most important applications of gas turbines is in power generation, though it has been in use for aircraft propulsion since long time.

The efficiency and power output of gas turbine plants is dependent on the maximum temperatures attained in the cycle. Advanced gas turbine engines operate at high temperatures (1200–15000C) to improve thermal efficiency and power output. With the increase in temperatures of gases, the heat transferred to the blades will also increase appreciably resulting in their thermal failure. With the existing materials, it is impossible to go for higher temperatures. Taking into account the metallurgical constraints,it is necessary to provide cooling arrangement for turbine blades to keep their metal temperature with in allowable limits. Therefore, developments in turbine cooling technology play a critical role in increasing the thermal efficiency and power output of advanced gas turbines. The following three types of cooling methods have been adopted to varying degree of success. 1. Convection cooling 2. Film cooling 3.

Transpiration cooling .While all three methods have their difference, they all work by using cooler air (bled from the compressor) to remove heat from the turbine blade. Convection cooling works by passing cooling air though passages internal to the blade. Heat is transferred by conduction to the blade and then by convection into the air flowing inside of the blade. A large internal surface area is desirable for this method, so the cooling passages are generally provided with small fins. Impingement cooling which is a variation of convection cooling, works by hitting the inside surface of the blade with high velocity air.

This allows more heat transfer by convection then regular convection cooling. Impingement cooling is often used on certain areas of the turbine blade, like the leading edge with standard convection cooling used in the rest of the blade. The second type of cooling is film cooling. This type of cooling works by pumping cool air out of the blade through small holes in the blade. This air creates a thin layer of cool air on the surface of the blade, protecting it from the high temperature gases.

The air holes are most often located along the leading edge. One consideration with film cooling is that injecting the cooler air into the gas flow reduces turbine efficiency. The drop in the efficiency increases as the amount of cooling flow increases. The drop in efficiency however is usually compensated by the increasing overall performance produced by the higher turbine temperature. Transpiration cooling is the third major type of cooling and is similar to film cooling as it creates a thin film of cooling air on the blade, but it is different in that that air is lead though a porous shell rather than injected through holes. This type of cooling is effective at high temperature asit uniformly covers the entire blade with cool air. The present paper attempts to study the effect of variation in number of cooling passages on the maximum temperatures attained and thermal stresses induced.The analysis is carried out under steady state conditions using Ansys software. The study has been conducted with three different classes of Nickel based alloys i.e., Inconel 718 and N-155 and chromium steel.

2. LITERATURE SURVEY:

Brahmaiah andLava Kumar [2] investigated the heat transfer analysis of gas turbine with four different models consisting of blade with without holes and blades with varying number of holes (5, 9&13) . The analysis was carried out using commercial CFD software FLUENT (a turbulence realizable k-ϵ model with enhanced wall treatment) has been used. On evaluating the graphsdrawn for total heat transfer rate and temperature distribution, the blade with 13 holes is considered as optimum. Steady state thermal and structural analysis was carried out using ANSYS software with different blade materials of Chromium steel and Inconel718.While comparing these materials Inconel718 is better thermal properties and induced stresses are lesser than the Chromium steel Chandrakant et. al [3] compared the performance of helicoidal ducted blade cooling with turbulator of different geometric proportion. It was found from analysis that there was significant improvement in cooling characteristics for turbine blade with turbulator geometry having larger e/D ratio. Also it was found from his analysis, performance has been vastly improved for greater thickness of turbulator geometry. e = radial thickness of turbulator rib D = outer diameter of helicoidally duct e/D is non dimensional ratio chosen as parametric variable for different configurations of rib geometries.

It was seen that an innovative helicoidal cooling passage, provides an augmented convective area for better heat dissipation. The helicoidal path also acts as a turbulence generator resulting in extended heat dissipation rates due to the geometry. The diameter and pitch length of the helicoidal duct plays a major role in optimizing the geometry of the helicoidal cooling passage. It was also found from analysis that e/D ratio of 0.08 and turbulator thickness of 0.75 mm provides best geometric configuration with respect to better heat dissipation characteristicsDeepanraj et al [4] used Finite element analysis to analyze thermal and structural performance due to the loading condition, with material properties of Titanium- Aluminum Alloy. Six different models with different number of holes (7, 8, 9, 10, 11, and 12) has been analyzed in his work to find out the optimum number of holes for good performance.

In Finite element analysis, first thermal analysis followed by structural analysis was carried out. Graphs are plotted for temperature distribution for existing design (12 holes) and for 8 holes against time. 2D and 3D model of the blade with cooling passages were shown. Using ANSYS, bending stress, deflection, temperature distribution for number of holes have been analyzed. It was found that when the numbers of holes are increased in the blade, thetemperature distribution falls down. For the blade configuration with 8 holes, the temperature near to the required value i.e., 800ºC was obtained. Thus a turbine blade with 8 holes configuration has been found to be the optimum solution.From the literature review, it observed that the cooling of gas turbine blades is a major consideration because they are subjected to high temperature working condition. Several methods have been suggested for cooling of blades as mentioned above literatures and one such technique is to have radial holes to pass high velocity cooling air along the blade span. the forced convection heat transfer from the blade to the cooling air will reduce the temperature of the blade to allowable limits, also it is observed that the combination of deferent types of cooling in the same blade are effective for high cooling.

2.1 PROBLEM STATEMENT:

With the advent in Gas turbine technology, its usage as a prime mover has become prominent, since last few decades. One of the most important applications of gas turbines is in power generation, though it has been in use for aircraft propulsion since long time.

The efficiency and power output of gas turbine plants is dependent on the maximum temperatures attained in the cycle. Advanced gas turbine engines operate at high temperatures (1200–15000C) to improve thermal efficiency and power output. With the increase in temperatures of gases, the heat transferred to the blades will also increase appreciably resulting in their thermal failure.With the existing materials, it is impossible to go for higher temperatures. Taking into account the metallurgical constraints,it is necessary to provide cooling arrangement for turbine blades to keep their metal temperature with in allowable limits.Therefore, developments in turbine cooling technology play a critical role in increasing the thermal efficiencyand power output of advanced gas turbines. New internal heat transfer data is needed to improve current rotor blade cooling performance. And also detailed flow and heat transfer data is necessary to understand the flow physics and to improve the current internal cooling designs.

Many techniques have been developed to enhance the heat transfer in these passages. The cooling passages located in the middle of the airfoils are often lined with rib turbulators. Although the techniques used to cool the blades are similar to those used to cool the vanes, the heat transfer trends in the vanes and blades are very different. Because the blades are rotating, the flow of the coolant in the passages is altered. Therefore, the effect of rotation on the internal heat transfer enhancement must be considered.

3.0.METHODOLOGY :
3.1 THEORETICAL CALCULATIONS:

Theoretical calculation has been divided into towcalculation ,heat transfer calculation and hydrodynamic calculation . some parameters have been estimated by using the following correlations.

$$Re = V * D * \rho/\mu \quad, \quad Nu=3.66 \quad, Nu = h \times D/k ,$$

$$V(r)=v_{max}(1-(r/R)^2), \quad V_{max}=2*v_{ave} \quad \Delta P = f \times (L/D)(\frac{\rho v^2}{2}) .$$

$$f=64/Re , \quad P(z) = P1 - (\Delta P/L) \times z , \quad Cf = f/4,$$

$$Le = 0.06 \times D \times Re .$$

The parameters are Reynold number ,Nusselt number, heat transfer coefficient, pressure profile, velocity profile, skin friction factor, hydrodynamic entry length. All parameters have been estimated in the cooling holes. Fig. 1 illustrates the RESULTS FROM THEORETICAL CALCULATIONS.

Results for Heat Transfer	Re	Nu		H.T.C (W/K m^2)
	621	3.66		168.543
Hydrodynamic Results for Velocity Profile	Vmax (m/s)	V(r) (m/s)		
	60	V(r) = 60*(1-(r/0.0005)^2)		
Hydrodynamic Results for Static Pressure Profile	P1 Pa	P2 Pa	ΔP Pa	P(z) Pa
	3337	0	3337	P(z)=3337-(3337/0.117) * Z
Hydrodynamic Results for Skin Friction Factor and Entry Length	Cf	Le (m)		
	0.0257	0.03726		

Fig 1 the RESULTS FROM THEORETICAL CALCULATIONS.

NOMENCLATURE
Symbol   Description

Nu   Nuselt Number

h   Heat Transfer Coefficient [W/m^2 K]

V_{ave} Inlet Velocity of the Hole [m/s]

θ   Non-Dimensional Temperature

δ Non-Dimensional Span Length of the Blade

zdistance along the span length where the blade temperature is measured (m)
Z Overall Span Length (m)
TbBulk Temperature (K)
dACross Sectional Element (m^2)
u Local Velocity (m/s)
valvalidation
C.D.F   Coefficient of Drag Force
C.L.F   Coefficient of Lift Force
ΔTTemperature Difference (K)
qhsurface heat flux (W/m^2)
ThTemperature of Hot Air[K]
TsTemperature of Blade Surface [K]
KhThermal Conductivity of Hot Air [W/m K]
L   Chord Length of Blade [m]
TCTemperature of Cooling Air [K]
 Re Reynold number of the cold air at the inlet of hole
 D Diameter of the Hole   [m]

3.2.Creating the CAD Model:

Models of blade (4 models with without holes and varying numbers of holes 4,8,12 holes) have been created using CATIA v5 software .fig3 shows blade model with 4 holes creating in CATIA by using aerofoil coordinates as shown in fig .2

X	Y	Z
48.5	0.5	0
45	3.95	0
38.2	8.77	0
26	13.6	0
21.1	14.9	0
16.18	15.5	0
11.2	14.5	0
6.18	12.4	0
3.2	13.5	0
2.6	17.3	0
5.85	21.5	0
10	25	0
14.8	26.6	0
22.9	25.3	0
24.5	24.7223	0
28	23	0
33.4	19.5	0
38	15.3	0
42	10.9	0
45.4	6	0
48.5	0.5	0

FIG.2 the airofoilcoordinatesof gas turbine blade.
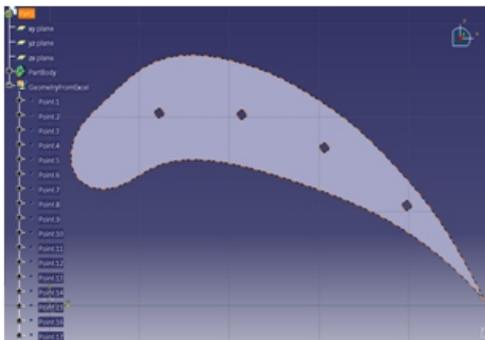
Fig 3: Create   blade model with four holes

3.3 Modelling of Geometry:

The next step involves the pre-processing part of CFD analysis by using Ansys workbench version 14.5. Among the pre-processing part the first step involves the geometry modelling in accordance with the compatibility of CFD analysis. Fig 4 shows the compatible model for the CFD analysis with boundary wall details.
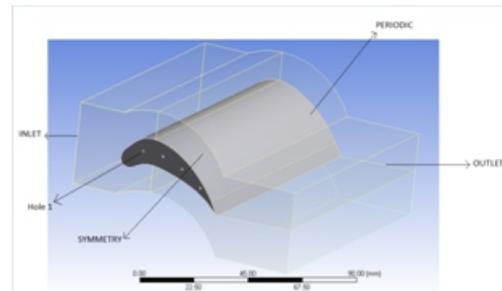

Fig 4: Geometry model with boundary walls.

3.4   Meshing of the Geometry

Meshing is defined as discretizing the domain into finite element model The Steps involved in the meshing process as per the figure 3-7 is as follows .Select mesh > mesh control > method. In the method dialogue box select all geometry and give the method option as sweep method. Select manual source and target option. Select symmetry surfaces of one side as the source and opposite side symmetry surfaces as target. Select the free face mesh type as Quad/tri. Define the number of divisions as 117.Select mesh > mesh control > sizing. In the face sizing dialogue box select the symmetry faces and define the element size as 1mm.Select mesh > mesh control >sizing.In the edge sizing dialogue box select the outer edges of the symmetry surfaces except edges at inlet and outlet. Define the element size as 1mm.Select mesh > mesh control >sizing. In the edge sizing dialogue box select the outer edges of aerofoil and the edges at the split of air domain near the trailing edge of the turbine blade. Define the element size as 0.2mm.Select mesh > mesh control > sizing. In the edge sizing dialogue box select the edges at the holes of turbine blade. Define the number of divisions as 40.Select mesh > mesh control >inflation.In the inflation dialogue box select geometry as the symmetry surfaces except the surface at the turbine blade at one end. Select the edges of air domain at the interface air and blade. Select the inflation option as total thickness. Number of layers as 27, Growth rate as 1.2 and maximum thickness as 2 mm respectively. Select mesh > mesh control > inflation In the inflation dialogue box select geometry as the exposed surfaces of the cooling air domains. Select the edges of cool air domain at the interface air and blade. Select the inflation option as first layer thickness. First layer height as 0.003 mm as per the Y plus equal to 1 which is the already proved value for the turbine blade meshing. Growth rate as 1.2 and maximum layers as 10 respectively. Fig 5 illustrates the mesh of the computational model of gas turbine blade .
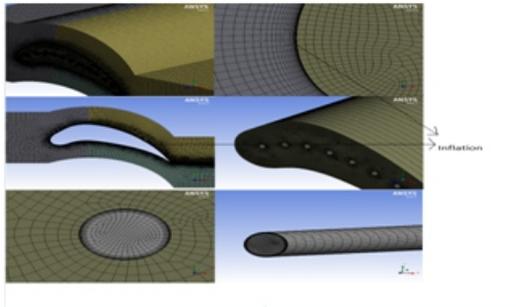
Fig 5 : Meshed Model

3.5 CFD solver setup:

The next step involved in this project is CFD solver setup and simulation of the flow. The software used for the turbine blade simulation is ANSYS Fluent version 14.5.The steps followed for the solver setup is as follows Open the Setup option of ANSYS fluent project. Select Solver setup > General > among solver settings choose the pressure based solver. CFD fluent consist of two solvers for solving the CFD process i.e. pressure based and density based. Here we chooses pressure based; this is because the fluid under consideration is air at subsonic speed, where the compressibility effects are small and can be neglected. Choose the velocity formulation as absolute and time option as steady state.

This is because flow is considered to have constant properties with respect to time. Select Solver setup > Models > Choose energy equation on. Turbine blade analysis is aiming mainly at the temperature distribution over the turbine blade. To solve the heat transfer problems the basic energy equation has to be used and this can be used by switching on the energy equation. Select the Realizable K – epsilon model with enhance wall treatment as the viscous model. This is because the hot air velocity is sufficient to make the flow turbulence over the airfoil.

The Standard k−ε is a well-established model capable of resolving through the boundary layer .Select solver setup > Materials > select fluid and click edit. Edit the properties of the fluid with the properties given the table 3-1 click change create. Select solver setup > Materials > select solid and click edit. Edit the properties of the solid with the properties given the table 1 click change create. Select Cell zone conditions and choose the appropriate materials for the zones like blade, hot air and cold air.

Select laminar for the zones of cold air. Click the operating conditions and verify that the operating pressure is 101325 Pascal. This is because we are assuming that this process is simulated at the atmospheric pressure at sea level.Select the boundary conditions and define the boundaries with appropriate input data According to our CFD model the hot air with temperature of 1112.22 K entering at a velocity of 265 m/s .Similarly the cold air at room temperature of 573 K is entering at a velocity of 30 m/s [1].Upper and lower surfaces of the air domain is defined with periodic boundary condition. Side surfaces of the full domain is defined with symmetry boundary condition.

This symmetry and periodic boundary conditions are inevitable to modify the analysis in such a way that infinite number of blades are arranged side by side. Interfaces walls between the blade and the air is defined with coupled wall boundary condition in order to activate the conjugate heat transfer between the blade and the air. In the solution method choose pressure velocity coupling scheme as SIMPLE. , SIMPLE is the acronym of semi implicit method for pressure linked equations.

It is the default scheme and the robust one which can be used for all types of analysis. Choose momentum, turbulent kinetic energy and turbulent dissipation rate and energy as second order upwind. Second order upwind is the best discretization method for this project since the mesh is made with tetrahedral elements and the flow is not aligned with the grid. Select the monitors > residuals. Click edit.

In the dialogue box that appears edit the convergence criteria of continuity, x velocity, y velocity, z velocity, energy, k, epsilon as 1*10-6. Close the dialogue box. In the solution initialization select the hybrid initialization and click initialize. By starting CFD simulation we must provide fluent with an initial guess for the solution flow field.

Hybrid initialization is a collection of recipes and boundary interpolation methods. It solves Laplace's equation to determine the velocity and pressure fields. All other variables, such as temperature, turbulence, species fractions, volume fractions, etc., will be automatically patched based on domain averaged values or a particular interpolation recipe.

Materials	Air (hot air)	Air1 (cooling air)	Chromium steel	Super alloy Inconel 718	N155
Density kg/m³	0.615	Incompressible ideal gas	7750	8190	8249
Specific heat J/Kg K	1047	1164	435	586	435
Thermal conductivity W/m K	0.04605	0.074025	24	25	20
Viscosity Kg/ms	2.971*10⁻⁶	4.5505*10⁻⁵	-	-	-

Table 1 Physical Property Details of Fluid Used In CFD Analysis

4.0 RESULTS AND DISCUSSION:

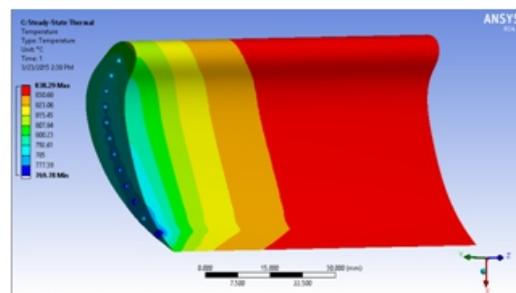Fig.6 Distribution of temperature around the surface of blade without holes

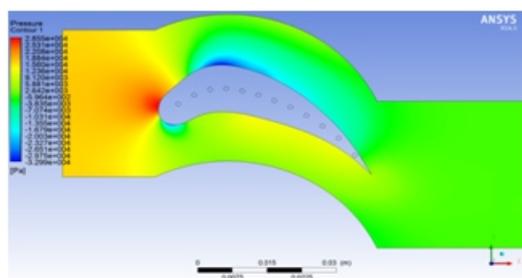

Fig.7 Distribution of pressure of main stream around the 12-holes Inconeel -718 blade
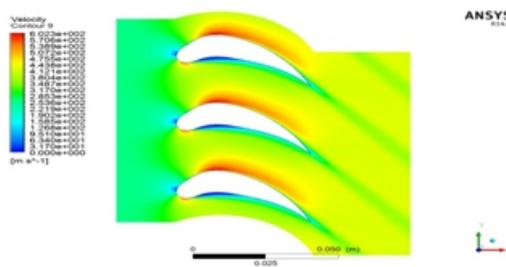
Fig.8 Distribution of velocity of main stream around the12 -holes Inconel-718 blade
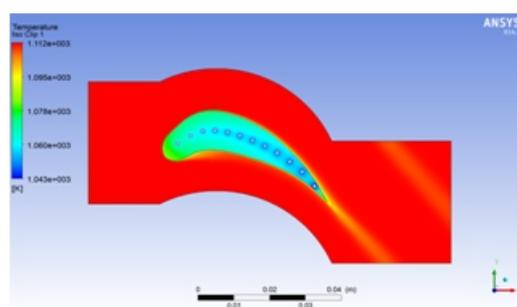
Fig. 9 Distribution of a temperature around and inside the 12 - holes Inconel-718 blade

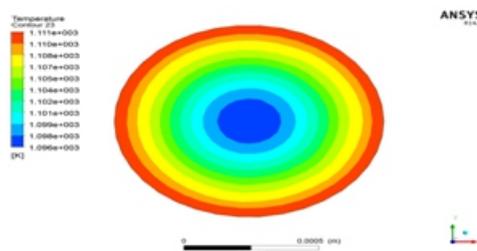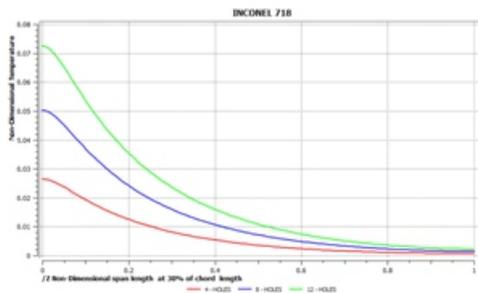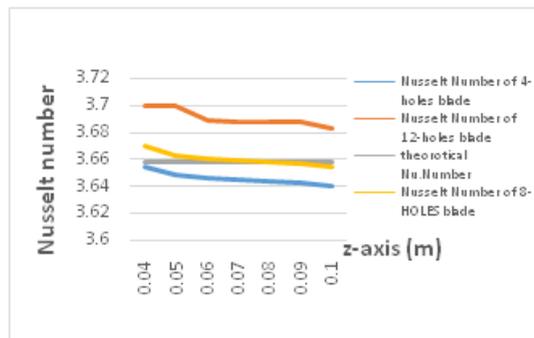Fig.10 Temperature distribution for Inconel -718 material with 12-holes
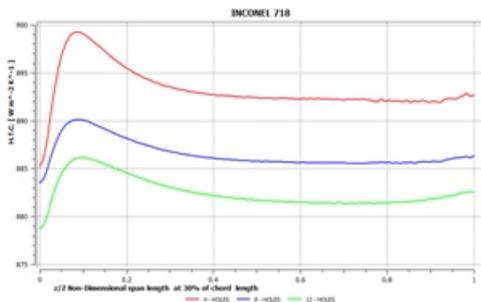

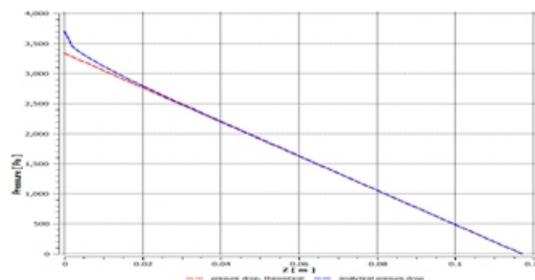Fig.11 Distribution of temperature at the outlet of hole1 for 12 - holes Inconel -718blade

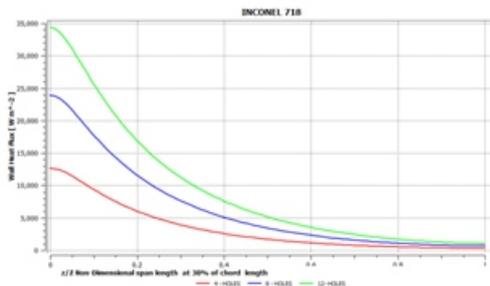**GRAGH 1   N.D.T Variation around the S.L.B at 30 % of the chord length for Inconel 718 blade material**

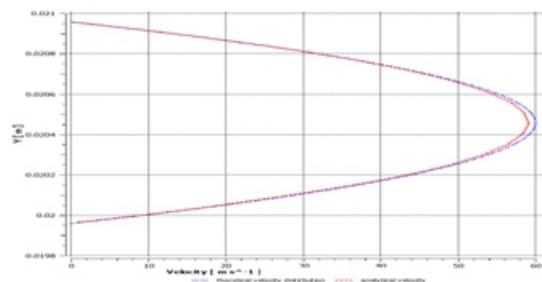**GRAGH 2   H.T.C. Variation around the S.L.B at 30 % of the chord length for Inconel-718 blade material**

GRAGH 3   W.H.F Variation around the S.L.B at 30 % of the chord length for Inconel 718 blade material

GRAGH4 Theoretical and Analytical Skin Friction Factor along the span length of holes

GRAGH5 Surface Nusselt Number along the surface of hole 1 for Cr-St blade with 4,8 and12- holes

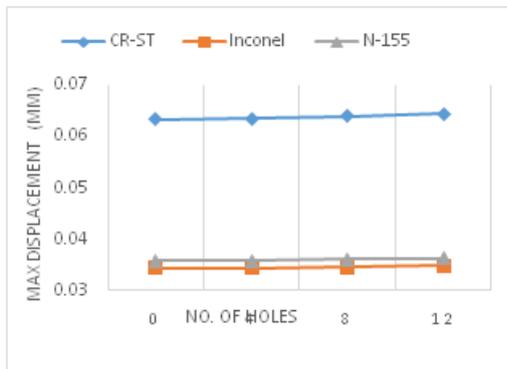**GRAGH6   Theoretical and Analytical Pressure distribution along the span length of holes**

GRAGH7   Theoretical and Analytical velocity distribution along y-axis of holes atHydrodynamic Entry Length
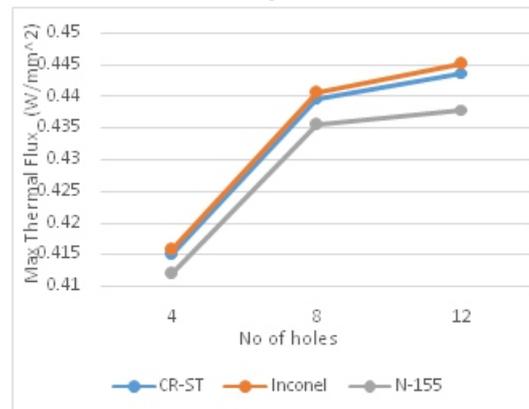
GRAGH 8   Max. Thermal Flux variation Vs. Number of holes

GRAGH 9   Max. Deflection Vs. Number of holes

z (m)	Tb K	Tw K	Φ W/mK	H W/m²K	Nu
0.04	998.10	1101.3	17580.3	170.361	3.699
0.05	1029.7231	1104.4	12697.6	170.033	3.692
0.06	1052.5673	1106.58	9178.44	169.93	3.690
0.07	1069.0808	1108.15	6635.63	169.842	3.688
0.08	1081.0181	1109.26	4796.39	169.832	3.687
0.09	1089.6444	1110.05	3464.56	169.784	3.686
0.100	1095.8706	1110.59	2496.74	169.622	3.683

TABLE 2 Bulk Temperature, Wall Temperature, Wall Heat Flux, H.T.C and Nuselt Number variation along the surface of holes for INCONEL 718, 12 - holes blade

DISCUSSION:

In Fig.6 ,It can be observed that the temperature around and within the solid blade (blade without holes) is same for the inlet main stream. this is because of using steady state condition in the CFD analysis and the another reason is using incompressible ideal   gas for the hot air . It can concluded that there is no   heat transfer coefficient between the hot air and the surface of blade because of no heat flux. The heat transfer coefficient is derived from the temperature gradient. No temperature gradient, no heat transfer. The temperature gradient comes from the temperature field, which is the solution of the energy equation.

The energy equation consists of the diffusion terms (the conduction in fluid), the source terms (the heating element), and the convection terms (coupled through the velocity field). Therefore, in order to solve the energy equation for the temperature field, one needs the velocity field. The velocity field comes from the solution of the continuity equation and the three momentum equations.By observingfigures (7 and 8   ) related to pressure and velocity distribution   of mainstream around the blade with 12 holes forInconell-718 , The maximum   static pressure is increasing with increasing the number of holes. Similarly, the maximum velocity is decreasing with increasing number of holes. By using the incompressible ideal gas for hot air, when the surface of blade is cooled, the density of air becomes high and the velocity decreases, so according to Bernoulli's principle, the pressure increase with decrease velocity.

The minimum velocity has been obtained with 12-holes blade for all material. When approaching to the leading edge of a blade, the hot air first slows down and then it speeds up again as it passes over or beneath the blade. As the velocity changes, so does the dynamic pressure and static pressure according to Bernoulli's principle. Air near the stagnation point has slowed down, and thus the static pressure in this region is higher than the inlet static pressure to main duct. Air that is passing above and below the blade, and thus has speeded up to a value higher than the main inlet path velocity, will produce static pressures that are lower than inlet static pressure. At a point near maximum thickness, maximum velocity and minimum static pressure will occur.

Figures (9 and 10) indicate to distribution of a temperature within the blade and distribution of a temperature around the surface of blade. This Figures show that the hole1 is high wall temperature because the effect of mainstream flow is high at leading edge. also they show the last hole is low temperature because of the small distance between the suction and pressure side and slightly reduced temperature at the surfaces of blade resulting in cooling of main hot air during passing through the blade surface. It can be observed from the above figs that the temperature at the root of the blade is lower and it increases towards the tip of the blade. This characteristic can be explained from the fact that the cooling air is at its lowest temperature (300 C) at the root of the blade. While flowing through the cooling holes from the hub to the root of the blade, the cooling air goes on increasing along the radial direction.

It is observed that the leading edge temperature for the blade consisting of high number of holes is lesser than the blade consisting of low number of holes. From above figures, it can be observed that the trace of reduced temperature of the main stream is near the trailing edge. That means the temperature of mainstream is reduced .this causes a decrease of turbine efficiency. Figure 11 indicates to the distribution of temperature at the suction of the outlet of holes for Inconel 718material with 12 holes blade. The minimum temperature is obtained at the center of this suction of holes in addition to the temperature at the center of holes is lower for blade with 12-holes than for blade with 4 and 8 holes. This occurs because of the effect of cooling holes with each other's leading to reducing in temperature at the outlet of the holes.

From figureis mentioned above, it is observed that the outlet temperature is same for all blades having the same number of holes. This characteristic can be explained from the fact that when the flow of air passes through the holes, it will reach to the region called thermal fully developed. At this suction of holes, the temperature of stream remains constant even itmoves to the end of holes. The temperature of cooling air at the outlet of holes is less than the main stream of hot air, so the airhave ability to absorb more heat from the hot main stream of air through the blade. The action of absorbing can be occurred by making the flow of cold air as turbulent flow by increasing the velocity of cooling air and diameter of holes moreover it could make the ribs inside the holes to make flow as turbulent.

From Graph 3 related to 12 holes Inconel blade material. It is observed that the heat flux for blade with 12-holes is greater than of blade with 4 and 8-holes .In addition to , from Graph 2 ,the H.T.C for blade with 4-holes is greater than the blade with 8 and 12 holes respectively . This is because the temperature deference between the main hot air and the surface of the blade is increasing with increasing the number of holes compared with slightly increasing of heat flux passing through the blade. Let us consider that quantity of heat passing through the surface of blade is constant. By increasing the number of cooling holes, the temperature difference between the surface of the blade and mainstream increases and then the heat transfer coefficient decreases .it is described the above phenomena by using the Forriers law of cooling as shown below [18].

H.T.C=qh/ΔT =qh/(Th- Ts), Where, H.T.C is heat transfer coefficient around the span length of blade at 30% of chord length of blade ,Khis thermal conductivity of hot air , L is the chord length of the blade. Graph 1 related to 12-holes for Inconel blade materials shows the non-dimensional temperature of the exposed blade surface (θ) along the span length at 30% chord length (δ).The non-dimensional temperature (θ) is defined as:

θ= ((Th- Ts))/(Th- TC), Where,Th Is hot gas temperature surrounding the blade profile ina convective ambience.Ts Is the surface temperature along the span length of the blade..TC Is temperature of cooling air admitted to the cooling holes at the blade root. Similarly, the non-dimensional span length (δ) is defined as:

δ =z / Z ,Where zis the distance along the span length where the blade temperature is measured.Z is overall span length..

Form table 2 , hole 1 is taken for analyzing of blade models. It is observed that bulk temperature along the sections of the hole for all materials is decreasing with increasing the number of holes at the specified region of the blade section and it increases along the span length of the hole 1. This is because more heat absorbed by cold air when it moves through hole from root to the tip of blade. The bulk temperature is defined as below.

$$Tb = \frac{rate\ of\ enthalpy\ through\ a\ cross\ section}{rate\ of\ heat-capacity\ through\ a\ cross\ section}$$

$$Tb = \frac{\int_A \rho\, u\, cp\, T\, dA}{\int_A \rho\, u\, cp\, dA}, \quad WheredA\ is\ cross\ sectional\ element,\ \rho\ and\ \boldsymbol{u}$$

are the density and local velocity [13].u = u(x, y) of the flowing fluid respectively. Graphs 5 illustrate the variation of analytical and theoretical NusseltNumber along the span length of hole1 (z).It is observed that the value deference between them is low. The validation of the analytical solution can be obtained by below correlation by selecting hole1 for CR-ST material with 4- holes:

$$val = \frac{theorotical\ nusselt\ number - analytical\ nusselt\ number}{theorotical\ nusselt\ number},$$

$$val = \frac{3.66-3.65}{3.66} = 0.0016,\ Wherever\ is\ the\ validation\ of\ analysis$$

The same method can be applied for calculation the validation of skin friction factor, max velocity and velocity distribution at the hydrodynamic entry length. The steady state thermal and structural analysis is carried out with different blade materials of chromium steel,

Inconel 718 and N-155 to determine the thermal flux and stresses induced in the blade. . It can be observed from Graph 8, the thermal flux for 12 holes is higher side for Inconel718 material compared to chromium steel and N-155. . It can be observed from graph 9 that the deformation is higher for the chromium steel material compared to Inconel-718 and N-155. Inconel material has minimum value of deformation and is within allowable limits. Hence, it is concluded that blade consisting of 12 holes has the maximum heat transfer rate, and minimum value of deformation is the best blade that can be used as gas turbine blade.

CONCLUSION:

In this thesis, a turbine blade is designed and modeled in CATIA v5 software. The turbine blades are designed using cooling holes. The turbine blade is designed with no holes, 4 holes, 8 holes, 12 holes. The present used material for blade is chromium steel. In this thesis, it is replaced with Inconel 718 and N-155. CFD analysis, Thermal and Structural analysis is done in Ansys.By observing the CFD analysis results, the velocity of main stream(hot air) is decreased from no holes to 4 holes, 8 holes and 12 holes .But the value of pressure is slightly increased.

The total heat transfer rate is maximum and the temperature of the leading edge is minimum for the blade consisting of 12 holes for Inconel -718. The temperature of the surface of blade with 12holes for Inconel-718 is minimum. It is found that the temperature leaving the trailing edge is low leading to decrease in thermal efficiency of gas turbine. Result showed that heat transfer coefficient and Nusselt number on the surface of holes are nearly constant for all blade materials with different numbers of holes also the heat transfer coefficient is high at entrance region.

By observing the thermal analysis results, the thermal flux is more for Inconel 718 than the other materials. The strength of the Inconel 718 is also more than that of Chromium steel and N-155. The analyzed stress values for Inconel 718 are within the permissible stress value, and displacement values are less by observing the structural analysis results.So it can be concluded that using Inconel 718 is better. However, the main disadvantage of using is its weight since its density is more than that of chromium steel and N-155.

References:

1-A.Hasanpour, M. Farhadi and H.R. Ashorynejad , Hole Configuration Effect on Turbine Blade Cooling ,World Academy of Science, Engineering and Technology vol 49 2013.

2-K HariBrahmaiah, M.Lava Kumar ,Heat Transfer Analysis of Gas Turbine Blade through Cooling Holes , International Journal of Computational Engineering Research (IJCER) 2014.

3-Chandrakant R Kini , Computational Conjugate Heat Transfer Analysis of HP Stage Turbine Blade Cooling Effect of Turbulator Geometry in Helicoidal Cooling Duct , World Academy of Science, Engineering and Technology Vol:6 2012 .

4-B. Deepanraj, Theoretical Analysis of Gas Turbine Blade By Finite Element Method ,Scientific World, Vol. 9, No. 9, July 2011.

5-G Narasa Raju , Steady State Thermal & Structural Analysis of Gas Turbine Blade Cooling System ,International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 1, January- 2013.

6-Josin George , The Design And Analysis Of First Stag Gas Turbine Blade With A modification Cooling Passage Using Ansys ,InternationalJournal of Latest Trends in Engineering and Technology (IJLTET) 2014.

7-KamleshBachkar,Static And Thermal Anlysis Of Turbine Blade OfTurbocharger ,Technological Institute, Mumbai, Maharashtra, ndia.© 2014 IJEDR | Volume 2, Issue 2 | ISSN: 2321-9939.

8-M P.V.Krishnakanth , Structural & Thermal Analysis of Gas Turbine Blade by Using F.E. International Journal of Scientific Research Engineering & Technology (IJSRET) Volume 2 Issue2 pp 060-065 May 2013.

9-A.Immarigeon, An advanced impingement/film cooling scheme for gas turbines – numerical study, International Journal of Numerical Methods for Heat & Fluid Flow 2005

10-ShridharParegouda , CFD Simulation on Gas turbine blade and Effect of Hole Shape on leading edge Cooling Effectiveness, International Journal of Modern Engineering Research (IJMER), Vol. 3, Issue. 4, Jul - Aug. 2013 pp-2066-2072.

11-Tutorial: Solving A Conjugate Heat Transfer Problem Using ANSYS FLUENT

11-Module A: Laminar Pipe Flow By ANSYS FLUENT.

12- Computing surface integral, Fluent Incorporation.

13- Arnal, M.P, " A General Computer Program For Tow-Dimensional Turbulent, Re-   Circulating Flow ",Report No.Fm-83-2,1983.

14-Launder, B.E. and Spalding, D.B.," Mathematical Model Of Turbulence ", Academic press, London, 1972.

14-Karki, K.,andPatankar, S., "Pressure Based Calculation Procedure For Viscous Flows At All speeds in arbitrary configurations", ALAA Journal, Vol. 27,PP. 1167-1174,1989.

15-Verestage, H. K., and Malalasekera, W., "An Introduction To Computational Fluid Dynamic-the finite volume method ", Longman Group Ltd,1995.

16- Fluent Inc., Fluent User' s Guide, Version 6.3.26, 2009.

Author's Details:

Aqeel Jomma Athab
ME Student,
Department of Mechanical Engineering,
JNTUH College of Engineering,
Hyderabad-India.

Dr.S.Naga Sarada
Professor,
Department of Mechanical Engineering,
JNTUH College of Engineering,
Hyderabad-India.