Advanced Cooling of Gas Turbine Blades with Sodium Liquid and Air

Jovin Jose, Dr. V. V Prathibha Bharathi

Department of Mechanical Engineering, Malla Reddy College of Engineering (MRCE), Secunderabad, India

Abstract— *Gas turbines are extensively used for air craft propulsion, land based power generation and industrial applications. Thermal efficiency of gas turbine can be 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 technique that describes in thesis is by cooling the blade by circulating sodium liquid filled inside it. The hot less denser sodium liquid travels the bottom region of the blade and transfer heat to the high velocity air passing near it. In this thesis a turbine blade model is designed and modeled in CATIA V5. CFD analysis of the model is performed for variant models. A comparative study of the results like temperature, turbulent kinetic energy and heat transfer rate through cooling air is made to find the best model which delivers effective cooling of the blade.*

Keywords— Ansys Fluent, Blade Cooling, CFD, Optimization

I. INTRODUCTION

A turbine blade is the individual component which makes up the turbine section of a gas turbine. The blades are responsible for extracting energy from the high temperature, high pressure gas produced by the combustor. The turbine blades are often the limiting component of gas turbines. To survive in this difficult environment, turbine blades often use exotic materials like super alloys and many different methods of cooling, such as internal air channels, boundary layer cooling, and thermal barrier coatings. Turbine blades are subjected to very strenuous environments inside a gas turbine. They face high temperatures, high stresses, and a potentially high vibration environment. All three of these factors can lead to blade failures, which can destroy the engine, and turbine blades are carefully designed to resist those conditions.

Turbine blades are subjected to stress from centrifugal force (turbine stages can rotate at tens of thousands of revolutions per minute (RPM)) and fluid forces that can cause fracture, yielding, or creep failures. Additionally, the first stage (the stage directly following the combustor) of a modern turbine faces temperatures around 2,500 °F (1,370 °C), up from temperatures around 1,500 °F (820 °C) in early gas turbines. Modern military jet engines, like the Snecma M88, can see turbine temperatures of 2,900 °F (1,590 °C). Those high temperatures weaken the blades and make them more susceptible to creep failures. The high temperatures can also make the blades susceptible to corrosion failures. Finally, vibrations from the engine and the turbine itself (see blade pass frequency) can cause fatigue failures. There are many types of cooling used in gas turbine blades; convection, film, transpiration cooling, cooling effusion, pin fin cooling etc. which fall under the categories of internal and external cooling. While all methods have their differences, they all work by using cooler air (often bled from the compressor) to remove heat from the turbine blades. The study of film cooling on a gas turbine rotor blade is performed by K. Takeishi, S. Aoki, T. Sato and K. Tsukagoshi. The film cooling effectiveness on a low-speed stationary cascade and the rotating blade has been measured by using a heat-mass transfer analogy. The film cooling effectiveness on the suction surface of the rotating blade fits well with that on the stationary blade, but a low level of effectiveness appears on the pressure surface of the rotating blade. Film cooling performance for injection through discrete holes in the end wall of a turbine blade is investigated by M. Y. Jabbari, K. C. Marston, E. R. G. Eckert and R. J. Goldstein. A. Immarigeon, (Department of Mechanical and Industrial Engineering, Concordia University, Montréal, Canada), I. Hassan, (Department of Mechanical and Industrial Engineering, Concordia University, Montréal, Canada) made a numerical study on advanced impingement/film cooling scheme for gas turbines. The Purpose of the present study aims to conduct a numerical investigation of a novel film cooling scheme combining in-hole impingement cooling and flow turbulators with traditional downstream film cooling, and was originally proposed by Pratt & Whitney Canada for high temperature gas turbine applications. The performance related to the heat transfer coefficient is a prospective topic for future studies. Practical implications are that advanced and innovative

cooling techniques are essential in order to improve the efficiency and power output of gas turbines. As per the research journals it is very clear that cooling of gas turbines are limited with the conventional methods where air is the only medium of heat transfer. Now a day's all conventional techniques of blade cooling are based on the same principal which is forced convection cold air which is passed through the channels or the blade carries out the heat. Basically this type of forced convection adversely affects the overall aerodynamic efficiency of the blade because of the interaction of the escaped cool air with the hot downstream air.

This thesis deals with a new ideology which utilizes two mediums for blade cooling.. The prescribed model uses circulated sodium liquid as the effective medium which transfers the heat to the cold air pumped through the fir tree section of the blade. The analysis is carried out under steady state conditions using Ansys Fluent software. The study has been conducted to identify the cooling effectiveness of the model by introducing porosity with different materials.
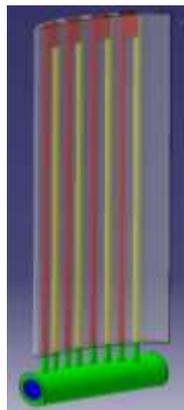
II.   MODEL DESCRIPTION

Fig. 1: Design Prototype

The sodium liquid filled inside the blade in a inverted U type pipe. The channel section of the inverted U tube facing the leading edge of the blade is coated with copper material. This is marked with a color of red in the Fig.1. The other channel is isolated with a small air gap clearance to prevent the heat transfer from the blade. This section is marked with color yellow. In the root of the blade a porous collector region is modeled exactly inside the fir tree. This is marked with the green color. There is channel passing through the porous collector region where cool air at high velocity bypassed from the initial stages of compressor is allowed to pass through. The sodium liquid extracts heat from the blade increases temperature and its density decreases. The lighter sodium liquid is allowed to pass to the porous collector region because of centrifugal acceleration which is prominent

when compared to acceleration due to gravity. The porous region with increased surface area and high turbulence transfer the heat to the cool air channel passing near the collector region. The cooled sodium liquid is allowed to pass through the insulated channel of the U tube because of high centrifugal acceleration. The cooled liquid again gets heated and process continues. We made a comparative study of the above model by adding and removing porosity in the regions of cooling channel and collector. Also we made a study by changing the material of porous region.

III.   METHODOLOGY

3.1. Turbine Blade Design Calculations

Input parameters of jet engine turbine blade is as follows

Mass flow rate = 20kg/s

Inlet Pressure P_{01} = 4 bar

Turbine Inlet Temperature (TIT) = 1100 ^oK

Axial Velocity (Constant through Stage) C_a = 260 m/s

Blade Speed U = 360 m/s

Rotational speed = 200rps

Nozzle efflux angle α_2 = 65 degrees

Stage exit swirl α_3 = 10 degrees

Assume nozzle loss coefficient = 0.05

Assume T_{01}-T_{03}= 145 ^{o} K

Assume C_p = 1148 j/kg K

All parameters of blade velocity diagram of Fig.3 have been estimated using the following correlations.

$\psi = (2Cp\Delta T_{0s})/U^2$, $\Phi = C_a/U$

$Tan\ \alpha_3 = tan\ \beta_3 - (1/\ \Phi)$

$Tan\ \beta_3 = (1/2\ \Phi)\ ((1/2)\ \psi + 2\ \lambda)$

$Tan\ \beta_2 = (1/2\ \Phi)\ ((1/2)\ \psi - 2\ \lambda)$

$Tan\ \alpha_2 = tan\ \beta_2 + (1/\ \Phi)$

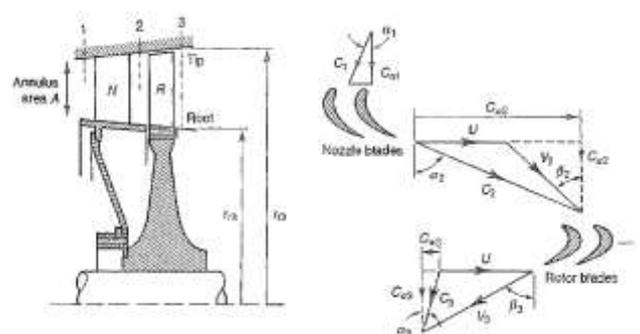$C_{a2} = U\ \Phi,\ \ C_2 = C_{a2}/Cos\ \alpha_2$
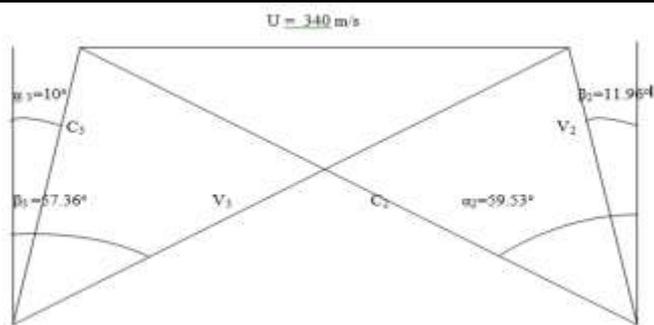

Fig. 2: Blade angles detailed view

Fig. 3: Velocity Diagram

Further the blade stage height and cross sectional area of air between the blades at different stages are calculated with the following correlations.

$T_0 - T = C^2/2C_p$, $T - T` = (\lambda_n C^2)/2C_p$

$p_0/p = (T_0/T`)^{\gamma/(\gamma-1)}$, $p_0/p_c = ((\gamma+1)/2)^{\gamma/(\gamma-1)}$

$\rho = p/RT$, $A = m/(\rho C_a)$, $A_N = m/(\rho C)$

$U_m = 2\pi N r_m$, $h = (AN)/U_m$

Hence the blade height and cross sectional entry area of air is tabulated in the TABLE.1

Table.1: Results of Blade Design Geometry

Station	1	2	3
A	0.0651 m^2	0.0864 m^2	0.1087 m^2
h	0.0478 m	0.0635 m	0.0799 m
r_t/r_r	1.25	1.34	1.45

The average height (h) of the station 2 and station 3 is taken as the blade height of the prescribed model.

3.2. Modeling the Geometry

Models of blade with cooling arrangement is created using CATIA V5 software.

The blade is model with the shape of NACA 2012 airfoil. The co-ordinates of airfoil are imported into the modeling software. The blade is modeled with a chord length of 22.3 mm length and an inclination of 10 degree which is calculated as the swirling angle α_3 in blade design calculations. The blade is made cambered in such a way that the trailing edges of the blade suites the angle β_3. The corresponding sketch is extruded and the blade is modeled as per the calculations. Blade cooling arrangements are then modeled with operations of pocketing, extrusion etc. The CATIA model is the saved in .stp format.
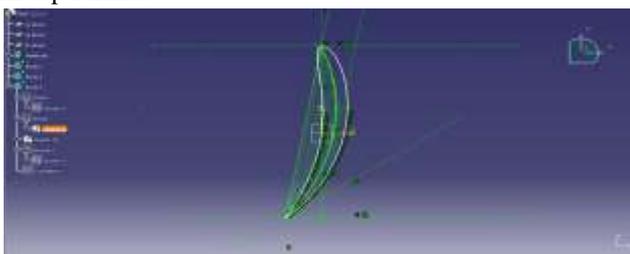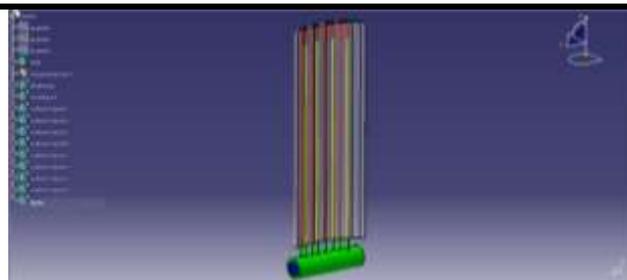

Fig. 4: Blade Sketch in CATIA

Fig. 5: 3D Model in CATIA

3.3. Meshing the Geometry

Meshing is defined as discretizing the cad model into finite element model. The meshing is done with Ansys Mesh module. The meshing of geometry is meshed with both tetrahedral and quadrilateral elements. Sweep method is used to mesh the blade and air domain. Face sizing of 0.2 mm is given to the source of sweep mesh. Inflation is defined on the fluid structure interaction region with 5 layers and growth rate of 1.2. The overall mesh quality is good and has a maximum skewness of 0.9 which is quiet acceptable for the analysis using Ansys Fluent solver. Faces are named using the 'Named selection' option of the mesh   to identify the boundary conditions properly.
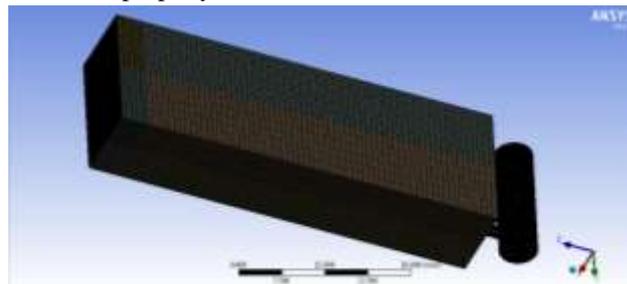

Fig. 6 : Meshed Model

3.4. CFD Solver Setup

Analysis of the model is performed using the solver Ansys Fluent. Ansys Fluent consist of two solvers for solving the CFD process i.e. pressure based and density based. Here we choose pressure based; this is because the fluid under consideration is air at subsonic speed, where the compressibility effects are small and can be neglected. We have selected 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. In the models choose the 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. We choose turbulence model for the analysis as standard K- omega .Standard K-omega model is one of the most commonly used turbulence models. It is a two equation model that means, it includes two extra transport equations to represent the turbulent properties of

the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy. The first transported variable is turbulent kinetic energy, k. The second transported variable in this case is the specific dissipation, omega. It is the variable that determines the scale of the turbulence, whereas the first variable, k, determines the energy in the turbulence. Specify the material properties in the materials. The properties of the materials are listed in the TABLE 2.

Table.2: Material Properties

Materials	Sodium Liquid	Air	Chromium Steel	Copper
Density kg/m^3	Polynomial function with two coefficients and variable as temperature. Developed from Boussinesq approximation $\rho = \rho_0(1 - \beta\Delta T)$ Coefficient 1 = 877.8 Coefficient 2 = -877.8×0.00024 = -- 0.2115498	Incompressible ideal gas	7750	8978
Specific heat j/kgK	1320	1006.43	435	381
Thermal conductivity w/mK	75.84	0.0242	24	387.6
Viscosity Kg/ms	0.0003878	1.7894*10^{-5}	-	-

In the cell zone conditions Blade is defined with chromium steel. Sodium liquid zone is specified with sodium liquid. Air domain is specified with air. Wick region is specified with sodium liquid. Cooling air is specified with air. Specify the porosity as 0.5 and material as copper or chromium steel. In air domain select the fixed value and specify the y velocity as -260 and click ok. In the operating conditions tab and verify the operating pressure as 101325 Pascal. Specify the gravitational acceleration g in z direction as 324982 m/s^2. Since the blade is rotating the gravitational acceleration is negligible when compared to centrifugal acceleration and is acting on the fluid domain as v^2/R. Given the blade is rotating at a speed of 200 revolutions per minute and equivalent radius is 0.206 m. According to our CFD model the hot air with temperature of 1100 K entering at a velocity of 260 m/s and passes over the blade .Similarly the cool air at a room temperature of 300 K is entering at a velocity of 160 m/s and passes through the cooling pipe fixed in the fir tree of the blade. Specify the Inlet boundary condition as pressure inlet with gauge pressure 0 Pascal and temperature as 1100 ^{o} K. Specify the outlet boundary condition as pressure outlet with gauge pressure 0 Pascal and temperature as 300 ^{o} K. Specify the

Inlet.cool boundary condition as velocity inlet with velocity of 160 m/s and temperature as 300 ^{o} K. Specify the Outlet.cool boundary condition as pressure outlet with gauge pressure of 0 Pascal and temperature as 300 ^{o} K. Most of Interfaces walls between the blade and the air are defined with coupled wall boundary condition in order to activate the conjugate heat transfer between the blade and the air. Especially the walls named as copper in the sodium liquid domain is specified with the material copper to increase the heat transfer property. Some interface walls of the sodium liquid domain is named as insulator walls are specified with heat flux 0 W/m^2. This is to modify the walls as the perfect insulator. 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 the gradient option as 'least square cell based'. It is the default method; has the good accuracy and the same time it is computationally less intensive. Choose PRESTO option for the pressure. It is used for highly swirling flows, flows involving steep pressure gradients. Choose momentum, turbulent kinetic energy and specific dissipation rate and energy as first order upwind. Specify the convergence criteria of continuity, x velocity, y velocity, z velocity, k, Omega as 0.001. Energy convergence criteria are 0.000001. Choose the hybrid initialization method to initialize the model. 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. In the Run calculation window edit the number of iterations as 10000 and click calculate. Within 2000 iterations solver will give the converged results with required accuracy.

IV.   RESULTS AND DISCUISSION

Results of the CFD analysis is extracted with the result module of workbench. Based on the results we made a comparative study on the four cases. They are

Case 1 - Without defining porosity for the domains collector and cooling air

Case 2 - Defining 50 % porosity only for cooling air domain with porous material as chromium steel (blade material)

Case 3 - Defining 50 % porosity for both domains cooling air and collector with porous material as chromium steel (blade material)

Case 4 - Defining 50 % porosity for both domains cooling air and collector with porous material as copper

4.1. Blade Temperature

The aim of the project itself is to decrease the temperature of the turbine blade to decrease the thermal stress acting on the blade. Therefore we made a comparative study on the blade maximum temperature of each case with the help of temperature contour. The sample view of the blade temperature contour is given in the Fig. 7
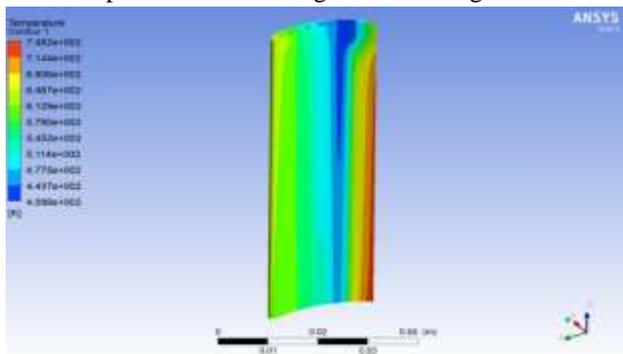

Fig. 7: Blade Temperature Contour of Case 4

Based on the temperature contours of all four cases we made a graph which plots the maximum temperature of the blade with different cases. The Fig. 8 shows the correponding graph of temperature contour.
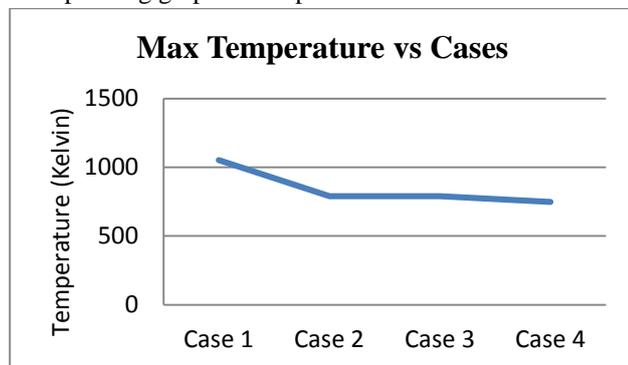

Fig. 8: Maximum Temperature against cases

From the Fig. 8 we can infer that the temperature of the blade is getting decreased from the case 1 to case 4. This is because of the effect of the porous region. Porous region increases the interfacial surface area of the fluid and solid which increases the convection heat transfer considerably. Similarly by replacing the porous material chromium steel with copper also increases the heat transfer rate because of good thermal properties of copper.

4.2. Turbulence

The principal difference between laminar and turbulent flow, as far as heat transfer is concerned is that an

additional mechanism of heat transfer in the radial and azimuthal directions become available in the turbulent flow. This is commonly termed 'eddy transport' and is intense, providing much better transfer of energy across the flow at a given axial position than in the laminar flow. Therefore more turbulence in the flow always gives more heat transfer rate. Generally the strength of turbulence is calculated with two parameters. They are turbulent kinetic energy and eddy frequency. Turbulence kinetic energy is the mean kinetic energy per unit mass associated with eddies in turbulent flow. Physically, the turbulence kinetic energy is characterized by measured root men square (RMS) velocity fluctuations. Eddy frequency is the rotation time for a typical turbulent eddy. When turbulence strength is more the eddy frequency is also very high. To identify why cooling effectiveness of the cases increases from case 1 to case 4 we made a study on the contours of turbulent kinetic energy and eddy frequency of the collector domain and the cooling air domain. The sample contours of turbulent kinetic energy and eddy frequency of collector and cooling air domain is shown in the Fig. 9, Fig. 10, Fig. 11, Fig. 12.
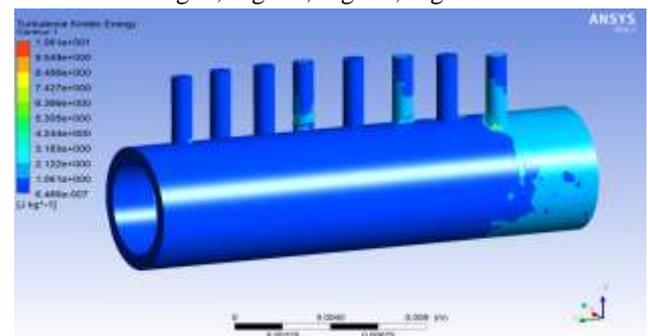

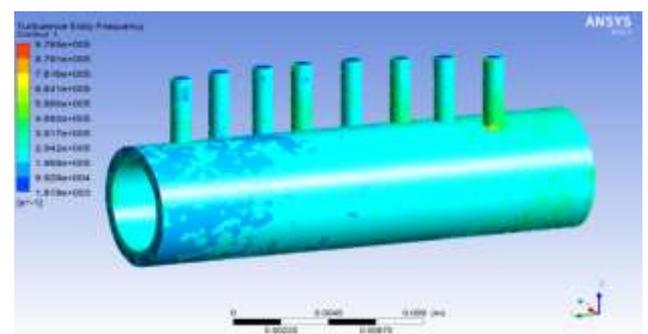*Fig. 9: Turbulence Kinetic Energy Contour of Collector for case 4*

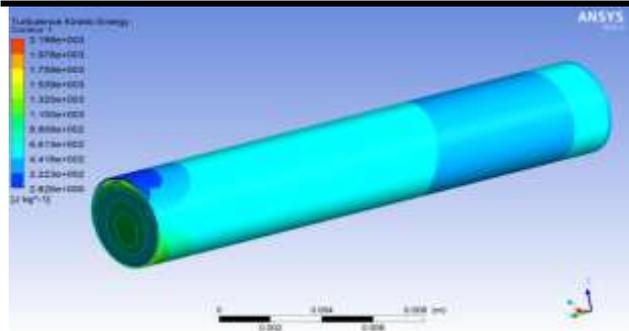*Fig. 10: Turbulence eddy frequency Contour of Collector for case 4*

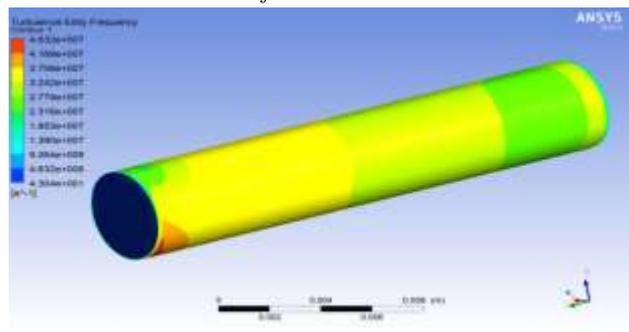*Fig. 11: Turbulence Kinetic Energy Contour of Cool air for case 4*

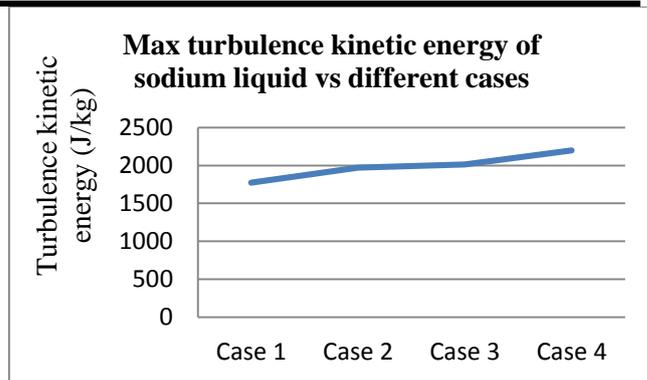*Fig. 12: Turbulence eddy frequency Contour of Cool air for case 4*

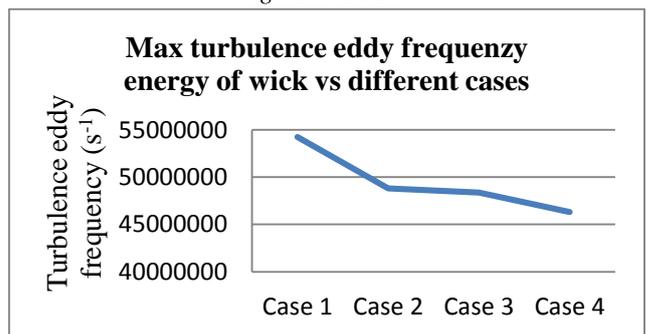*Fig. 13: Max Turbulence Kinetic energy of collector against Cases*

Fig. 14: Max Turbulence eddy frequency of collector against Cases

Fig. 15: Max Turbulence Kinetic energy of cool air against Cases

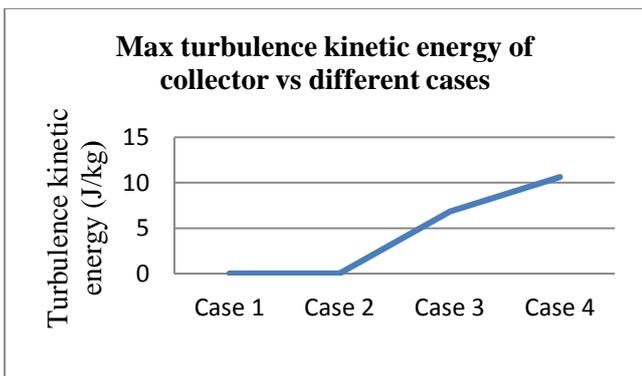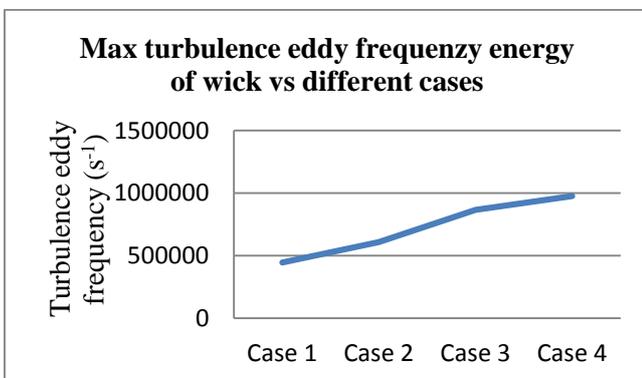*Fig. 16: Max Turbulence eddy frequency of cool air against Cases*

Based on the contours, graphs are made to compare the maximum value of turbulence kinetic energy and eddy frequency with different cases. These graphs are shown in the Fig. 13, Fig. 14, Fig. 15, Fig. 16 respectively. From the Fig. 13 and Fig. 15 the Turbulence kinetic energy is increasing from case 1 to case 4. This is because in former cases domains are defined without porosity and hence there is no obstacle for the flow and because of this the flow remains little turbulent. But while introducing porosity the turbulence increases because of the obstacles made with either chromium steel or copper and hence the secondary flow become predominenet. As a result turbulence kinetic energy of the flow increases. From the Fig. 14 and Fig. 16 we have to infer that the eddy frequency of the turbulence increases from the former cases to the later cases. When the energy of the turbulence increses the same amount of energy has to be dissipated shortly with the help of eddy formation. Larger the kinetic energy, larger the rate of energy dispation and so larger the rotataional frequency of the eddies.

4.3. Heat Transfer Rate

The heat carried over by the cooling mechanism is the amount of heat carried by the cooling air which is calculated by the basic equation $Q = m\,C_p\,\Delta T$. The heat transfer rate is calculated and tabulated.

Given C_p value of air = 1006.43 J/kg K

Table.3: Heat Transfer Rate by the Cooling Mechanism

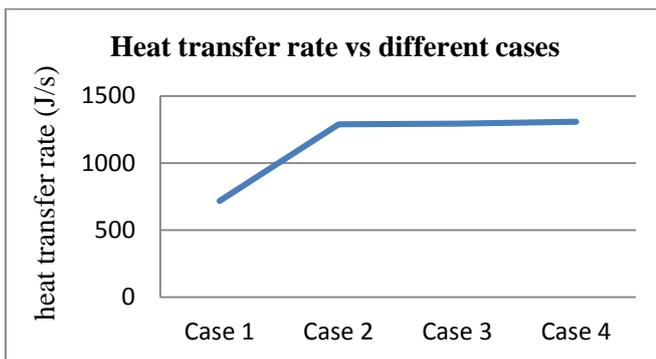| Case	m_{in} (kg/s)	m_{out} (kg/s)	T_{in} °K	T_{out} °K	$Q_{in} = m_{in} \times C_p \times T_{in}$ J/s	$Q_{out} = m_{out} \times C_p \times T_{out}$ J/s	$Q_{out} - Q_{in}$ J/s
Case 1	0.00274938	0.00274835	300	349.049	247.44	965.47	718.03
Case 2	0.00274938	0.00274814	300	555.821	247.44	1537.29	1289.85
Case 3	0.00274938	0.00275013	300	556.888	247.44	1541.36	1293.92
Case 4	0.00274938	0.00275094	300	561.826	247.44	1555.48	1308.04

Fig. 17: Heat Transfer Rate against cases

From the TABLE.3 and Fig. 17 we can infer that the heat transfer rate increases from the case 1 to case 4. This result is matching with the balde temperature where temperature of the blade decreaes from the case 1 to case 4.

V. CONCLUSION

The detailed comparative study of model with and without porous medium and different porous material is performed to find the most optimized and effective heat transfer. It is from the analysis we infer that porosity increases the heat transfer rate because of increase of the interface area of fluid and solid. Porosity also increases the turbulence due to obstacles made by porous region and an additional mechanism of heat transfer in the radial and azimuthal directions become available in the turbulent flow. One more feature which increases the heat transfer rate is the material which is used for porous region. Compared to the chromium steel, copper is having good thermal properties like thermal conductivity (which is approximately 4 times than that of chromium steel) which increases the rate of heat transfer. And finally this optimized design can replaces the conventional blade cooling techniques of forced convection through blades. Thereby eliminates the interaction of cooling medium with hot air and wont effect the aerodynamic efficiency of the blade adversely. The future scope of the project is to incorporate phase change in the region of free convection. The sodium liquid which is at pressure of the required boiling point as per the operating temperature of the blade undergoes the phase change process in which sodium liquid is changed to the state of vapor. The latent heat of vaporization utilized by the sodium liquid for phase change always maintains less temperature of the blade. The less dense vapor moves towards the wick and condenses by the cooling medium passes through the fir tree.

REFERENCES

[1] Film Cooling on a Gas Turbine Rotor Blade by K. Takeishi, S. Aoki, T. Sato and K. Tsukagoshi

[2] Film Cooling of the Gas Turbine Endwall by Discrete-Hole Injection by M. Y. Jabbari, K. C. Marston, E. R. G. Eckert and R. J. Goldstein

[3] An advanced impingement/film cooling scheme for gas turbines – numerical study by A. Immarigeon, (Department of Mechanical and Industrial Engineering, Concordia University, Montréal, Canada), I. Hassan, (Department of Mechanical and Industrial Engineering, Concordia University, Montréal, Canada)

[4] An experimental investigation on the trailing edge cooling of turbine blades by Zifeng Yang, Hui Hu

[5] CFD Simulation on Gas turbine blade and Effect of Hole Shape on leading edge Film Cooling Effectiveness by Shridhar Paregouda, Prof. Dr. T. NageswaraRao

[6] J.C. Han, S. Dutta, and S.V. Ekkad, 2000, "Gas Turbine Heat Transfer and Cooling Technology, "Taylor& Francis, Inc., New York, New York, ISBN # 1-56032-841-X, 646 pages.

[7] B. Lakshminryana , 1996,"Turbine Cooling and Heat Transfer, "Fluid Dynamics and Heat Transfer of TurbomachineryTurbomachineryTurbomachineryTurbomachineryTurbomachineryTurbomachineryTurbo machinery John Wiley, New York Wiley, pp. 597 – 721

[8] R.J. Goldstein 2001, "Heat Transfer in Gas Turbine Systems," Annuals of The New York Academy of Sciences, New York, Vol. 934,520 pages.

[9] R.E. Chupp, H.E. Helms, P.W. McFadden, and T.R. Brown, 1969,"Evaluation of Internal Heat Transfer Coefficients for Impingement Cooled Turbine Airfoils, "AIAA Journal of Aircraft6: pp.203-208

[10] R.S Bunker and D.E. Metzger, 1990,"Local Heat Transfer in Internally Cooled Turbine Airfoil Leading Edge Regions. Part I: Impingement Cooling without Film Coolant Extraction," ASME Journal of Turbo machinery 112: 451-458.

[11] Gas Turbine theory by Cohen and Rogers