International Journal of Engineering and Applied Sciences (IJEAS)
ISSN: 2394-3661, Volume-4, Issue-6, June 2017

Computational Investigation of Effect of Turbulator Arrangements on Turbine Blade Cooling

C. Sriram, S. Suthagar, M.R. Swaminathan

***Abstract*— Turbine inlet temperature is limited by metallurgical considerations and many modern engines make use of air cooled blades to permit operation at elevated temperatures and then it can be assumed that the mass flow remains constant throughout as explained earlier. At higher temperatures it is necessary to extract air from the compressor to cool both stator and rotor blades called as bleed flow of cooling air. The rotor blade cooling is the most difficult problem. It should not be forgotten that, with high gas temperatures, oxidation becomes as significant a limiting factor as creep, and it is therefore equally important to cool unstressed components such as nozzle blades and annulus walls. The objective of the analysis is to study the effect of reduction of temperature and to attain the maximum cooling efficiency on gas turbine blade cooling by varying the geometry of the cooling passages. An attempt is made in this paper to compare the performance of various shapes of blades by using computational fluid dynamics. It was found that blade cooling passage with angled turbulators is more effective in heat transfer while comparing with all other configurations of with and without turbulators.**

***Index Terms*— blade cooling, CFD, turbulator, turbulence**

I. Introduction

Gas turbines play a vital role in the today's industrialized society, and as the demands for power increase, the power output and thermal efficiency of gas turbines must also increase. One method of increasing both the power output and thermal efficiency of the engine is to increase the temperature of the gas entering the turbine. In the advanced gas turbines of today, the turbine inlet temperature can be as high as 1200°C; however, this temperature exceeds the melting temperature of the metal aerofoils. Therefore, it is imperative that the blades and vanes are cooled, so they can withstand these extreme temperatures. Turbine blade cooling plays a vital role in the performance of a gas turbine engine. The turbine inlet temperature usually exceeds the melting temperature of the metal airfoils. Therefore, it is imperative that the blades and vanes are cooled, so they can withstand these extreme temperatures. A number of traditional cooling concepts are used in various combinations to adequately cool the turbine vanes and blades. Computational methodology is effectively used to optimize the blade cooling. An interesting new technology that combines pin, impingement and film cooling in an attempt to further increasing the Rotor Inlet Temperature, is what this rapport will be about. It is referred as" IFC", standing for "Impingement/Film Cooling". The idea behind IFC is that instead of as for normally film cooling, where the fluid is ejected directly out to the surface and free stream air U_∞ , instead letting it pass a sophisticated flow passage[1]. Advanced cooling concepts with modification in shape and size were also investigated [2]. To improve turbine entry temperature for maximizing the thermal efficiency of the HP stage gas turbine blade, studies on performance of helicoidal ducted blade cooling with turbulator of different geometric proportion were carried out. It is found from analysis that there is significant improvement in cooling characteristics for turbine blade with turbulator geometry having larger e/D ratio [3].The mass transfer analogy has been used in literature to avoid the conduction related issues A less cumbersome non-contact mass transfer analogy based on Pressure Sensitive Paint (PSP). It has been possible to simulate actual engine density ratios using the PSP mass transfer analogy. The pressure sensitive paint is applied to the region of interest, which typically includes the region around and up to 30-40 diameters downstream of the film cooling holes [4]. Computational Fluid dynamics (CFD) is being used predict the location of possible damaged areas on turbine blades. These results could then be used as reference for carrying out non-destructive inspections. In this manner the number of blades inspected by per unit time could be substantially increased leading to savings in inspection cost, lesser repair time and more focused fault isolation in the blades [5]. Showerhead film cooling was found to augment Nusselt number and reduce adiabatic wall temperature downstream of coolant injection. The adiabatic effectiveness trend on the suction surface was also found to be influenced by a favourable pressure gradient due to Mach number and boundary layer transition region at all blowing ratio and exit Mach number conditions [6]. Double wall cooling uses a thin gap between two walls to enhance heat transfer from the surface of turbine blades. Double wall cooling increases area for heat transfer between cooling fluid and metal. Impingement jets and modified surfaces can be used to increase heat transfer on the outer wall [7]. Film cooling predictions are used to understand the mechanisms of the jets that exit these trenched holes and crater holes. The RSM (Reynolds Stress transport Model) for simulation of turbulent flows in film cooling and a simulation was run using FLUENT computer code. Comparisons made with experimental data for the film effectiveness distributions showed that the film cooling jet exiting the trenched hole is more two-dimensional than the typical cylindrical holes and crater holes [8].

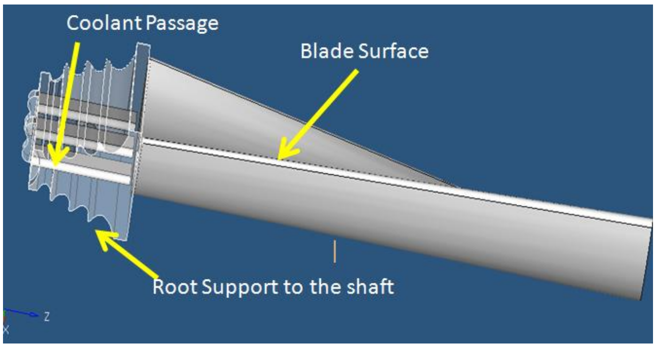
II. NUMERICAL METHODOLOGY

A. Physical Model

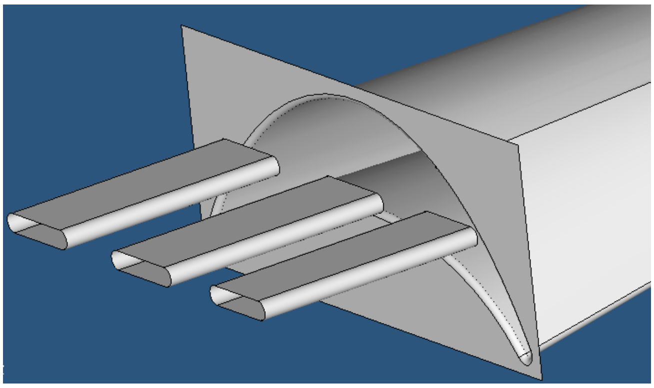
The 3D CAD model use for the present numerical investigation is shown in Figure 1. A twisted blade configuration is considered for the numerical simulation. The

C. Sriram, Department of Mechanical Engineering, Anna University, Chennai, India.
S. Suthagar, Department of Manufacturing Engineering, Anna University, Chennai, India.
M.R. Swaminathan, Department of Mechanical Engineering, Anna University, Chennai, India, 99625 87854.,

internal cooling passage at the root side of the turbine blade is also shown in Figure 1. The CAD model is prepared by considering straight cooling passages with and without turbulators. The turbulators are positioned in three ways such that cooling passages with single side oriented turbulators, double side oriented turbulators and right angled turbulators. The analysis for the present comparison is carried out with these three different turbulator positions. The base line model is considered as the turbine cooling passage without turbulators.



a) Basic Blade Profile



b) Cooling Passage at the Root of the Blade

Fig. 1. CAD model considered for present numerical investigation

Figure 2 shows the cooling passages with and without turbulators

B. Computational Domain

The turbine blade model is meshed with triangular elements on the surfaces and tetrahedral elements are created inside the fluid passage. Mesh with appropriate size is chosen to capture the turbulator shapes and cooling passages without any distortion in the flow domain.

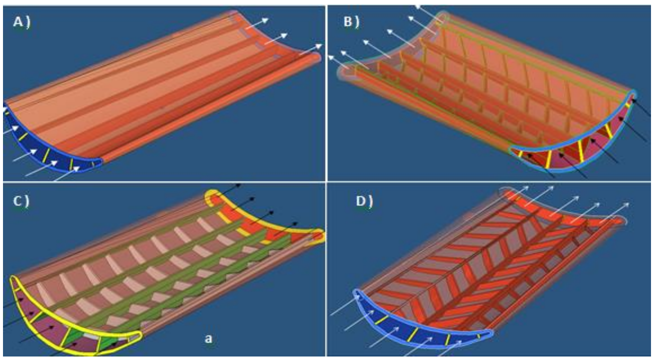


Fig. 2. Turbine Blade Model with Cooling Passages A) No Turbulators B) Angled Turbulators C) Orthogonal Turbulators D) V- Angled Turbulators

The surface mesh quality is maintained with a skewness of 60 degrees and the volume mesh is maintained with a tet-collapse within 0.9. Mesh refinements have been carried out wherever needed. Figure 3 shows the surface mesh of the turbine blade cooling passage with and without turbulators. The sectional view of the volume meshes indicating the solid turbine material and the coolant – fluid material along with V-angled turbulators. The tetra mesh growth rate is maintained as 1.2 from the surfaces.

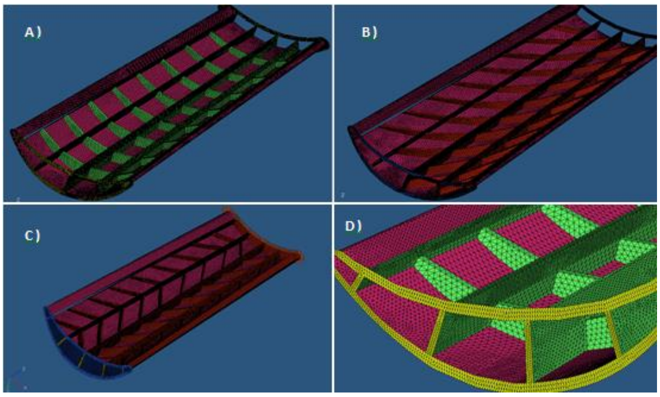


Fig. 3 Surface mesh at the Flow Passage of the Turbine Blade with Turbulators A) Orthogonal B) Angled C) V- Angled D) Mesh Refinement near the turbulators

Figure 4 shows another view of the volume mesh with V-angled turbulators

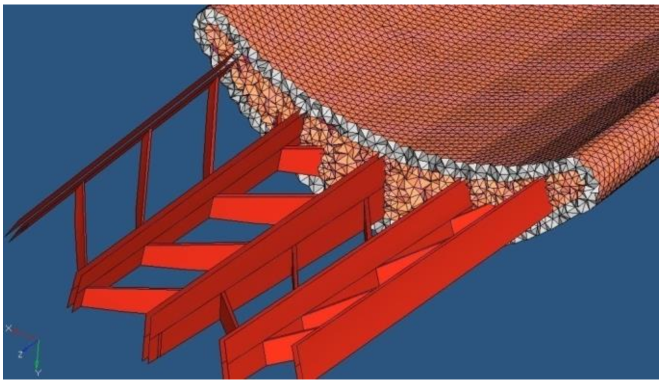
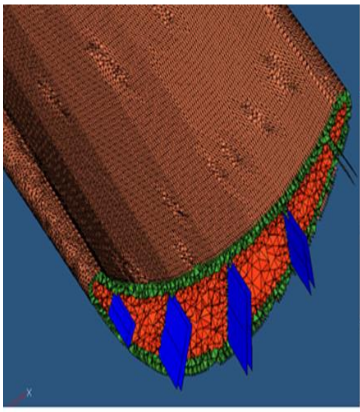


Fig. 4. Cut Sectional View of Volume Mesh – V-Angled Turbulators

The details of volume mesh across the blade are shown in the Figure 5. The volume mesh is done by using tetra mesh. The mesh details along the ducts and partition are seen in the figure.



Detail	Mesh Type	Count
Surfaces	Tri-elements	1479404
Volume	Tetra	676018
Nodes	-----	171834

Figure 5 Volume mesh and Mesh Count

C. Physics Definition

The numerical simulation on the discretized domain is carried out using ANSYS Fluent. Standard k-Epsilon turbulence model is used to predict the turbulence and viscous effects. A velocity specified inlet boundary condition is used at the coolant entry passage. The outlet is mentioned with 'Pressure Outlet' boundary condition. The medium level of turbulence intensity is specified at the inlet. The turbulators and other surfaces are specified with standard wall conditions with no-slip boundary conditions.

Bleeding air from the compressor is considered as the working fluid and the reference pressure is taken as 1 atm. The complete physics set up for the numerical simulation is given in Table I and Table II. A Pressure Based Navier-Stokes (PBNS) solver is considered for solving the continuity and momentum equations. The flow is treated as steady and incompressible. Nickel super alloy is considered for the blade materials and the properties are taken as Density = 8010 kg/m^3; Heat Capacity = 419 J/kg K and Thermal Conductivity = 10.9 W/m K.

Table I

Sl. No	Boundary	Boundary Type	Value
1	Fluid Inlet	Velocity Inlet	106 m/s
2	Fluid Outlet	Pressure Outlet	0 (g) Pa
3	Blade Surfaces	Wall –Temperature Specified	1100 K
4	Ducts / Turbulators	Wall- Coupled	No-Slip

Table II

Sl. No	Description	Solver Setting
1	Fluid Domain	Bleeding Air
2	Flow Type	Incompressible and Steady
3	Solver	3D-Pressure based Navier Stokes
4	Solid Domain	Nickel Super alloy
5	Turbulence Model	Standard k-Ɛ Equation
6	Equations Solved	Continuity , Momentum , Energy and Turbulence

III. RESULTS AND DISCUSSION

All the equations are solved with convergence criteria of 1x E-04 and the solution convergence is monitored. Figure 6 shows the convergence history of all equations in ANSYS Fluent

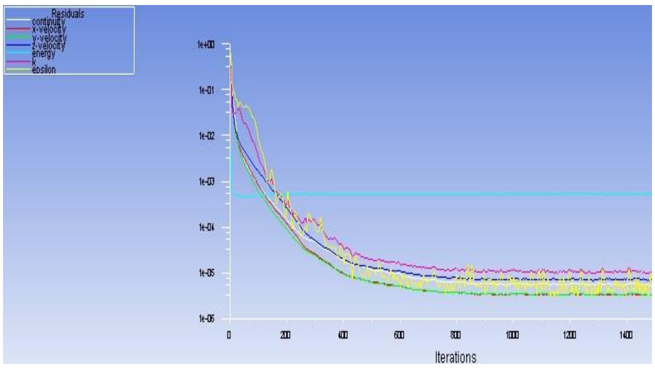


Fig. 6. Solution Convergence - History

Figure 7 shows the variation of temperature along the partition wall between the fluid and solid material. It is found that the effective cooling is only at the middle region of the coolant entry side.

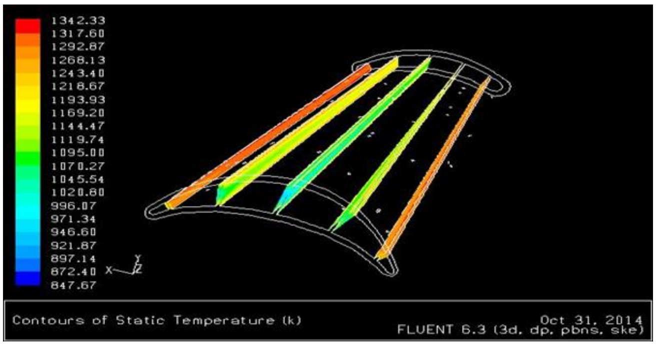


Fig. 7 Temperature Variation along the partition wall without turbulators

Figure 8 shows the variation of temperature along the cooling passage with orthogonal turbulators from which it is observed that the cooling effect is considerably increased due to the turbulators.

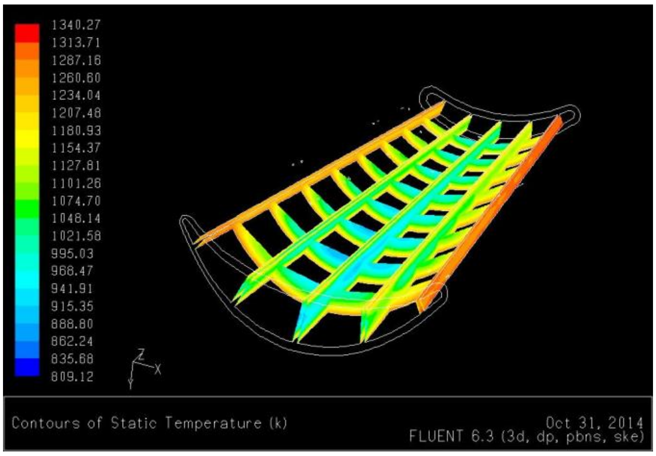


Fig. 8 Temperature Variation along the orthogonal turbulator passage

The temperature variation along the coolant passage with orthogonal and angled turbulators shown in Figures 8 and 9 indicates that the temperature distribution is effective in the middle passage while comparing the side passage.

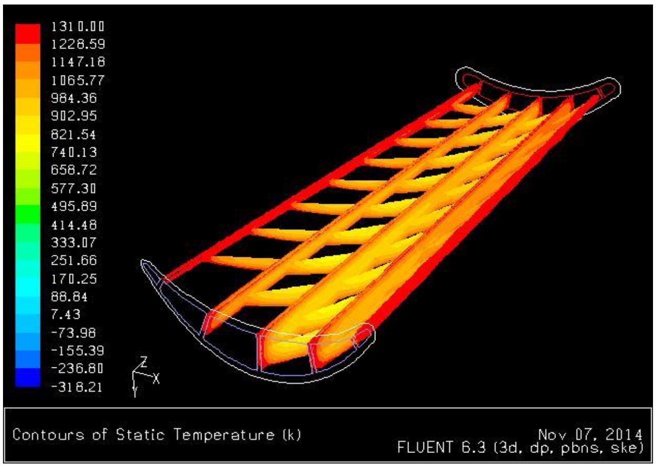


Fig. 9 Temperature Variation along the angled turbulator passage

Figure 10 shows the temperature variation along the coolant passage with V-angled turbulators. The lowest temperature is obtained at the entry of the coolant side.

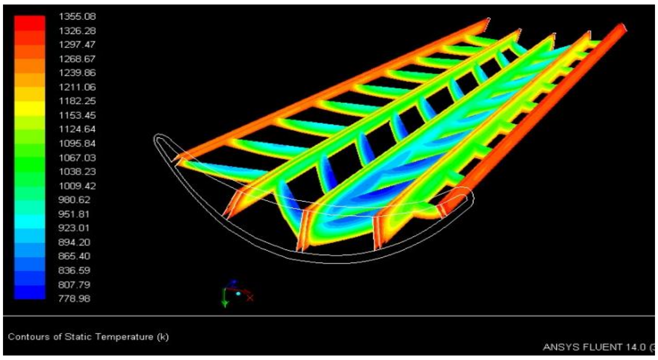


Fig. 10 Temperature Variation along the V- angled turbulator passage

Figure 11 shows the temperature variation along the coolant passage inner surface without turbulators. The vertical walls of the passage are cooled better than the bottom and top surfaces of the passage.

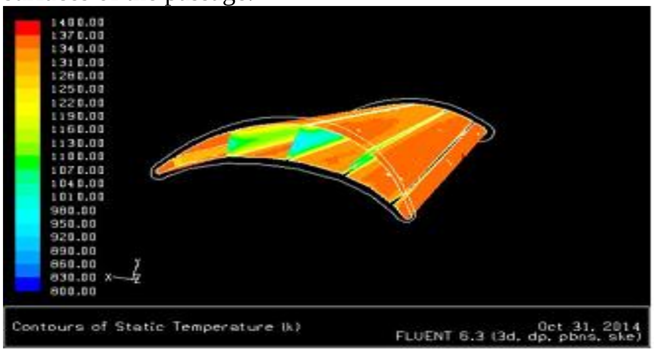


Fig. 11 Temperature Variation along the inner surface of the blade without turbulators

Figures 12 and 13 show the temperature variation along the coolant passage inner surface with Orthogonal, Angled and V-Angled turbulators. It is observed that the cooling effect is significant at the middle passage and also at the entry of the coolant side

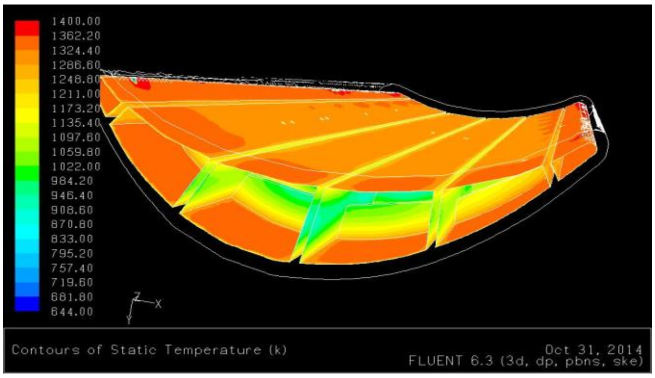


Fig. 12 Temperature Variation along the inner surface of the blade with orthogonal turbulators

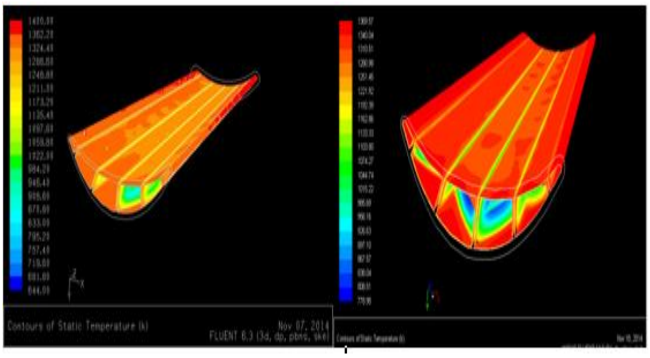


Fig. 13 Temperature Variation along the inner surface of the blade with angled and V-angled turbulators

Three different planes are considered from root to tip of the blade to show the variation of temperatures in cross wise planes alone the flow directions. Figure 14 shows the variation for the blade configurations without turbulators and also with Orthogonal, while Figure 15 shows angled and V-Angled turbulators.

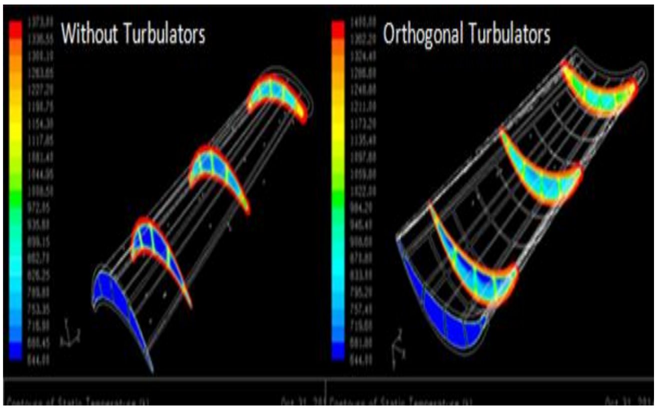


Figure 14 Temperature distributions in various cross planes along the flow – without turbulators and orthogonal

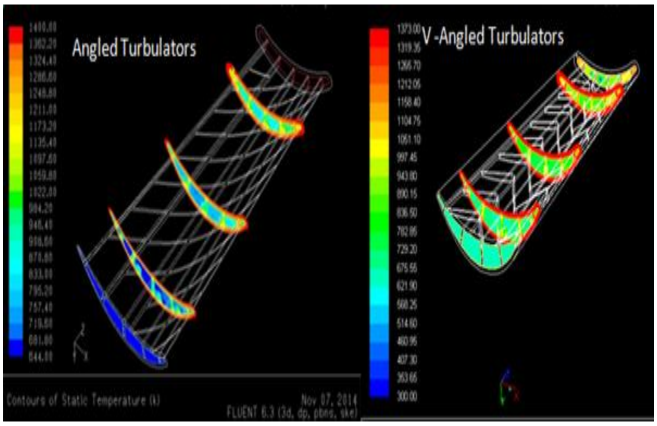


Figure 15 Temperature distributions in various cross planes along the flow – angled and V- angled turbulators

Figure 16 shows the temperature distribution from inlet to outlet at various planes.

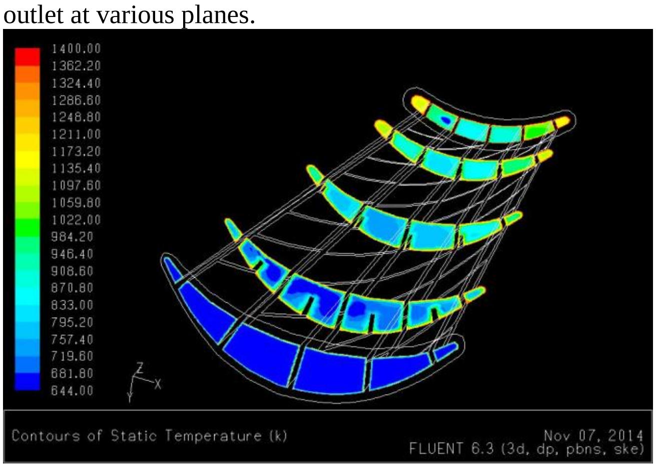


Figure 16 Temperature distributions from Inlet to Outlet at various planes along the flow (Angled Configuration)

Figures 16 and 17 shows the temperature variation over the inner surface of the blade configuration with V- Angled turbulators. It is observed that the CFD simulation predicts the temperature variation properly and there exist a significant cooling effect at the middle passage while comparing the leading and trailing edges of the blade configuration.

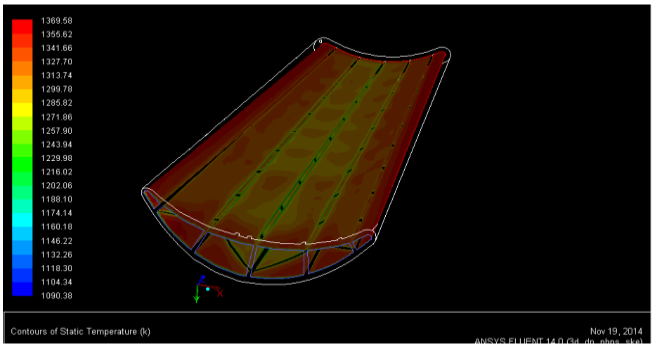


Figure 17 Variation of temperature along the inner surface of the blade for V- angled configuration

Figure 18 shows the temperature distribution along the inner surface of the blade for V-angled configuration.

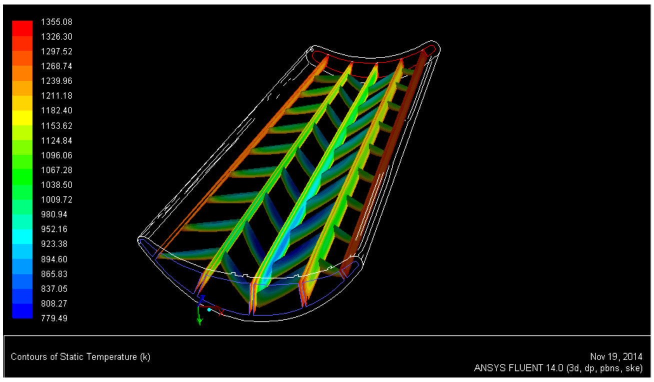


Figure 18 Variation of temperature along the inner surface of the blade for V- angled configuration

Figure 18 gives a comparison between the coolant outlet temperatures at the turbine blade exit for various configurations. It is observed from CFD results that the Angled turbulator configuration gives the better results while comparing with other configuration. Also it can be seen from the CFD values that the turbulators are more effective in increasing the heat transfer.

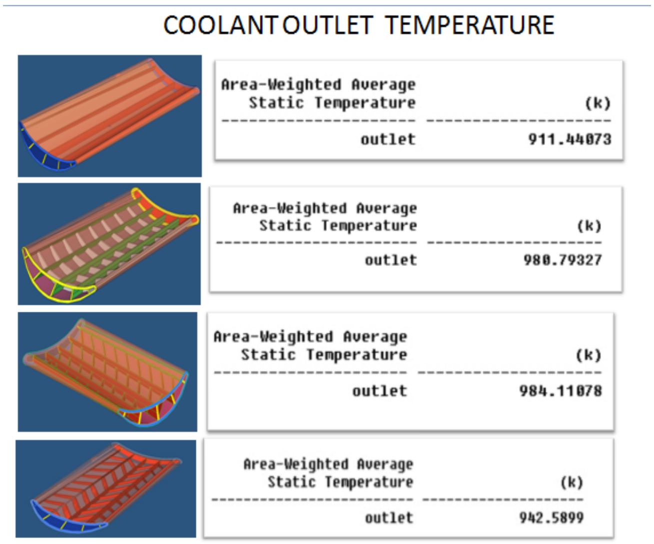


Figure 19 Comparison of various cooling passages

From the Table III it can be observed that the blade cooling passages with angled turbulators are more effective in heat transfer while comparing with all other configurations of with and without turbulators.

Table III Comparison of Heat transfer for various configurations

Configuration	**Average Heat Flux from Outlet (kW/m^2)**	**Net Heat Transfer Rate through Outlet (kW)**	**Area Weighted Average Temperature at Outlet (K)**
Passage without Turbulators	20.079	7.811	911.44
Passage with Orthogonal Turbulators	22.442	9.211	980.79
Passage with Angled Turbulators	32.022	9.259	984.11
Passage with V-Angled Turbulators	21.24	9.297	942.58

IV. CONCLUSION

Numerical Simulation of a typical gas turbine blade with various internal cooling configurations with and without turbulators has been carried out. CFD prediction of the temperature distribution along the ducts effectively and the cooling regions have been presented with appropriate contour plots. While comparing the temperature distribution across the blade, it is evident that overall comparison analysed from CFD results shows that the net temperature distribution as well as the net heat transfer rate taken by the cooing air is significantly more in the angled turbulator configuration while considering other three configurations. Hence it is concluded that the turbulators are more effective in heat transfer from hot gases to the coolant as well as the change of turbulator angle in uni-direction instead of bi-direction leads to better cooling.

ACKNOWLEDGMENT

The Author thanks Mr. G.Raju for providing the necessary software and graphic terminals to complete the simulation.”

REFERENCES

[1] Adrian Dahlquist,(2008), “Axial gas turbine blade cooling With Impingement/Film-Cooling”, Project Report Heat and Mass Transport, Lund University, Sweden.

[2] Erik Janke and Rolls-Royce plc., (2011), “Aero thermal Research for Turbine Components – An over-view of the European AITEB – 2 project”, Aero thermal Investigation on Turbine End walls and Blades.

[3] Chandrakant R Kini et al, SatishShenoy B and Yagnesh Sharma N, (2012), “Computational Conjugate Heat Transfer Analysis of HP Turbine Blade Cooling: Effect of Turbulator Geometry in Helicoidal Cooling Duct”, World of Science, Engineering and Technology 70 2012, pp. 645-652.

[4] Je –Chin Han, (2010), “Turbine Blade Film Cooling Frontiers using PSP Technique”, Frontiers in Heat and Mass Transfer (FHMT), 1,013001, DOI:10.5098/v1.1.3001, ISSN: 2151-8629, pp.1-21.

[5] M.Saqib Hameed and Irfan A Manarvi, (2011), “Using FEM and CFD for Engine Turbine Blades to Localize Critical Areas for Non Destructive Inspections”, International Journal of Multidisciplinary Sciences and Engineering, Vol. 2, No. 3, pp.29-37..

[6] Shakeel Nasir, (2008), “Showerhead Film Cooling Performance of Turbine Vane at high free Stream Turbulence in Transonic cascade”, PhD Thesis, Virginia Polytechnic Institute and State University, Blacksburg.

[7] Srinath V. Ekkad, (2010), “Advanced Internal Cooling Geometrics – Double walled Schemes with and without effect of rotation Heat”, Energy and Fluid Transport, Virginia Tech.

Yiping –Lu, (2007), “Effect of Hole Configuration on film Cooling from Cylindrical inclined holes for the application to gas turbine blade”, Ship building Institute, East China.