MODELING AND ANALYSIS TRAILING EDGE TURBINE BLADE COOLING

Anamika Kumari[1], Anil Verma[2]

[1]Research Scholar, Department of Mechanical Engineering, SOE, SSSUTMS, M.P., INDIA
[2]Assistant Professor, Department of Mechanical Engineering, SOE, SSSUTMS, M.P., INDIA

ABSTRACT

The gas turbine industry is always aim for increase the thermal efficiency of the gas turbine. One of the methods is by increasing the turbine inlet temperature. Increasing the operating temperatures may leads to some major problems. The melting temperature of almost materials is less than the design operating temperatures of the gas turbine. In addition to that the rapid spatial variations in temperature within the blade can create thermal stresses which can be in dangerous limits. In order to handle the problems related to thermal stresses, oxidation and creep which limit the lifetime of turbines, cooling of the blades is required.

Keyword: - *Gas Turbine1, Thermal Stresses2, Combustion Chamber3, and Turbines-Cooling4.*

1. GAS TURBINE OVERVIEW

A turbine is an indoor combustion engine that uses air because the operating fluid. Saravanamuttoo et al (2001) explain the conversion of energy from the fuel and also the operating fluid into energy within the production of power within a turbine. An example of turbine is painted in Figure 1 wherever the elements area unit visible through a cross sectional read. Careful clarification of the necessary elements of turbine is mentioned during this section.

Fig-1: SGT 750 Gas turbine

The turbine consists of 3 necessary components: the mechanical device, the combustion chamber, and also the rotary engine. Refer Figure 2 to envision the method of power generation within the turbine.

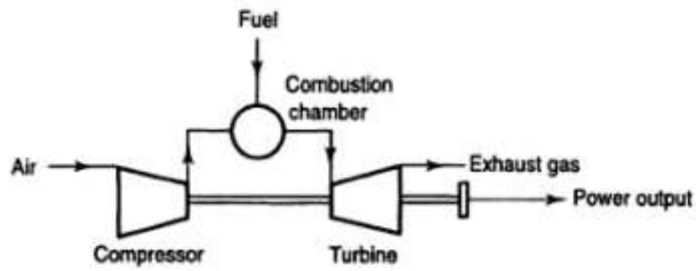

Fig-2: Simple Gas turbines Stage

The air is drawn from atmosphere pressure through the water guide vane to the mechanical device wherever the compression of water air takes place so, the combustion chamber comes into play wherever the fuel mixes with the water air to combust & move more to the rotary engine to get power.

1.1 Compressor Section

It is the responsibility of the mechanical device to supply aggressive flow to the combustion chamber with extreme potency. One stage of compression consists of a rotating blade that is linked to a rotating disk (the rotor), followed by a stationary vane (or stator). The flow space within the mechanical device blade & vanes area unit divergent. The Water & also the outlet guide vanes of the mechanical device area unit neither divergent nor focused. they 'reorganized to supply the simplest orientation for the mechanical device & combustor severally.

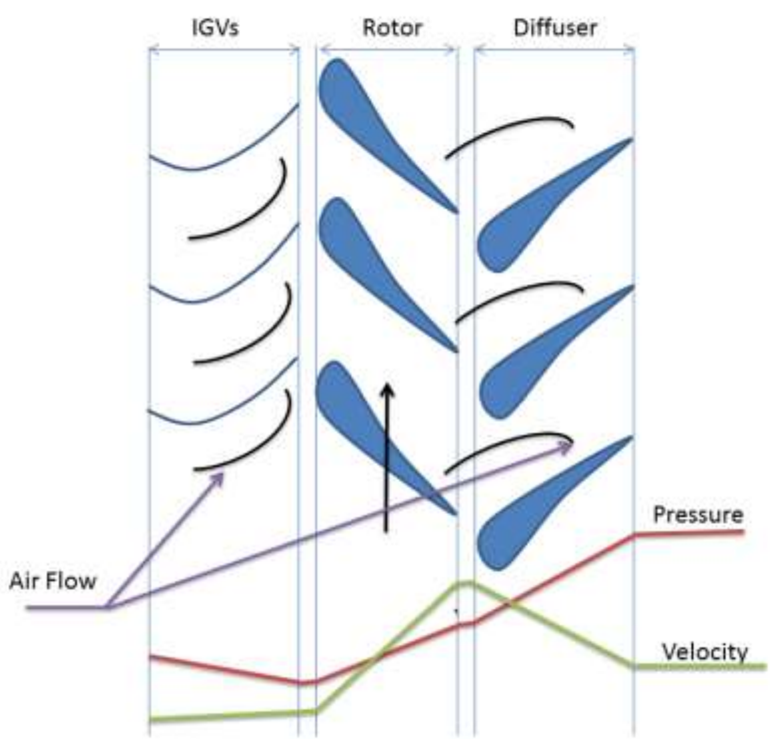

Fig-3: Compressor flow characteristics;

Figure 3 shows the pressure characteristics of the flow for one stage within the mechanical device section. The physical mechanism of the mechanical device is to convert the rotary energy in to the thermal energy. This conversion will increase the overall pressure (Pt) on that most of the rise is within the rate of the fluid with a little increase within the static pressure because of the divergence of the blade flow methods. The potency of the mechanical device depends on the smoothness of the flowing within the section & it's affected because of the occurrences of friction & turbulence. The air leaves the mechanical device through the diffuser into the combustion chamber. The Diffuser is important in changing the speed increase through the mechanical device section to static pressure. The Static pressure reaches the very best at the outlet of the diffuser.

1.2 Combustor Section

The task of the combustor section is dominant of the burning of the massive quantity of fuel & air in an economical manner. This should be achieved with minimum pressure loss & most heat unleash. additionally to it, the combustion should be oriented in such a way that heating of the metal components is avoided. The Combustion takes place within the primary zone or the face of the cans/burners. The first air is employed within the combustion method (approximately twenty five % of the water air) & also the remainder of the water air during this section is termed because the secondary air or the dilution air. The secondary air controls the flame pattern, cooling of the liner walls & will increase mass. The target is to accelerate the gas stream & reducing the static pressure providing a decent platform as water to the rotary engine section.

1.3 Turbine Section

The physical mechanism of the rotary engine section is to convert the thermal energy of the burnt fuel into energy. this is often achieved by increasing the new, aggressive gas from the combustor into lower temperature & pressure. The stator coil vanes increase the speed of the gas and also the rotor extracts the energy.

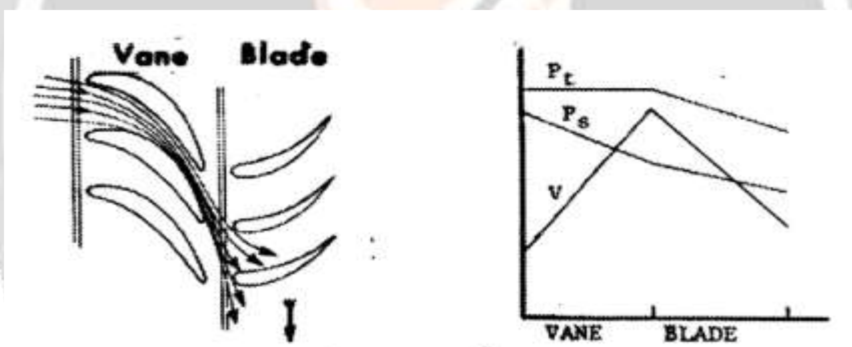

Fig-4: Turbine flow characteristics

The stator coil vanes area unit focused ducts which will convert the upper heat &pressure energy into higher rate gas flow. Velocity, temperature & also the pressure area unit sacrificed so as to rotate the rotary engine that successively generates the shaft power. The potency of this section is on most conversion of the energy from the new & pressure energy of the gases. The seal provided at the bottom & also the shroud accessible at the rear stages of the rotary engine contributes to the potency.

2. TURBINES COOLING

In trendy gas rotary engines cooled turbines area unit the primary stages of the turbine within which the cooling arrangements area unit provided. Cooling is important at this section as a result of the temperature is nearer to or bigger than the temperature of the fabric. In Figure 5, the primary 2 stages of the rotary engine have cooling arrangements & area unit connected to the mechanical device. this suggests that the work done by this rotary engine is employed to plot the mechanical device and so cited because the Compressor-Turbine (CT). Future2 stages area unit to blame for the transmission of the facility & cited as Power rotary engine (PT). The rotary engine water temperature is a crucial considers the look of the turbine & it's strived to keep up it as high

as potential to get most thermal potency. During a trendy turbine, the temperature is near 1500 degrees Celsius.
.

Fig-5: Turbine section after the combustion chamber

2.1 Turbine Cooling Design

The cooling style needs to bridge the gap between endlessly increasing gas temperature & also the pressures and also the allowable material temperature, that have raised considerably within the recent years. The gas temperature is influenced by the main target on the rotary engine water temperature & thermal potency.

2. LITERATURE SURVEY

S. Friedrichs [1] had developed a method to estimate the film cooling effectiveness. In their experiments, a thin film of 0.05 mm thickness of diazo surface coating was applied over the turbine cascades. Their cascade model consisted of four blades with a chord of 278 mm, span of 300 mm with the flow entering at 40°. The reactive nature of diazos towards ammonia would help to apply the heat mass transfer co-relations. The authors had compared the results against an experimental work conducted using a temperature based measurement techniques for film cooling. The results were in agreement apart from the regions near the coolant holes.

G Hyams [2] had studied the film cooling for critical parameters such as blowing ratio (1.25, 1.88), Density ratio (1.6) and length to diameter ratio (4). Their study included the experimental as well as CFD simulations. The authors had performed steady-state, 3-dimensionsal simulations using RANS approach. They had studied five film cooling hole geometries – cylindrical film hole, forward diffused film hole, laterally diffused film hole, inlet shaped film hole and cusp-shaped film holes. Based on their results, the authors had observed that the film cooling shape had a significant impact on flow distributions at the exit plane which could further influence the downstream film cooling performances. Among the five film cooling holes studied in this work, the authors noted that the laterally diffused film holes provide better effectiveness. This could be attributed to the presence of weak longitudinal vortices which ensures the coolant films to firmly attach to the surface. The film cooling effectiveness and the resulting heat transfer coefficient on a cylindrical leading edge was studied experimentally with the help of a low-speed wind tunnel, corresponding to the flow conditions of Re = 100,900.

Spinach V Ekkad [3]. The injection holes on the cylindrical specimen were arranged in rows at ± 15° from the stagnation. Their study focused on obtaining the film cooling effectiveness using liquid crystal technique. With the liquid crystal coatings on the cylinder surfaces, experiments were conducted to estimate the heat transfer coefficient and film cooling effectiveness. The cylindrical specimen was placed inside a tunnel with a cross-section of 25.4 cm X 76.2 cm. A suction type blower was used for supplying air which was heated by the heater. The cylinder surface was also heated by a set of six cartridge heaters. Based on their findings, the authors had concluded that the Nusselt numbers, hence the heat transfer rate, increases with the increase in blowing ratio.

K. T. McGovern [4] had conducted computational along with experimental investigations for studying flow mechanisms for the film cooling for various compound injection angles (45°, 60° and 90°). Their computational studies were conducted using ANSYS FLUENT. In order to limit the numerical viscosity in the CFD simulations, the higher-order linear reconstructive discretization scheme was selected by the authors in their simulations. They had used the Reynolds Averaged Navier Stokes (RANS) approach, with standard k-epsilon for turbulence closure, in their investigations. The authors had invoked symmetry approach in these simulations because of the geometrical and the expected flow field symmetry. The CFD simulations were observed to be in good agreement with the experimental data. They had concluded that the compound-angle injection method provides the lateral spreading of vortices in the domain, a significant.

The film cooling heat transfer rate and the associated aerodynamic losses were experimentally studied by J. E. Sargison [5]. The geometry considered by the authors was a flat plate with a converging slot hole, defined as console, for film cooling. The authors had compared the film cooling effectiveness for four different holes – slot, console, fan shaped and cylindrical – for pitch ratios. The experiment studies were corresponding to the flow conditions of Re = 144,000. From their study, the authors had observed that the console shaped film cooling method induces lesser aerodynamic losses as compared to the remaining cases. The laterally averaged heat transfer coefficient was higher in console and slot film cooling methods over the cylindrical and fan shaped film cooling holes.

Another research by Reaz Hasan [6] had focused on effusion film cooling for an adiabatic flat plate. Their study was conducted for various velocity ratios, defined as the ratio between the coolant velocity and free-stream velocity, ranging from 0.25 to 1.0 with the interval of 0.25. They had also studied for various hole arrangement as well. The authors had conducted computational simulations using ANSYS FLUENT. The steady-state pressure-based solver was chosen by them for the simulations. The turbulence closure was obtained by k-epsilon model, with the enhanced wall treatment to model the near-wall boundary layer effects. In their study, the authors had applied the incompressible ideal-gas approximation for modeling the Air. The pressure-velocity correction was obtained using the SIMPLE algorithm. They concluded that the effusion film cooling effectiveness increases with the increase in velocity ratio.

A three-dimensional CFD simulation based investigation to estimate the influence of multi-hole arrangement over the cooling film development was conducted by Yang Chengfeng [7]. They had studied the arrangement of holes – square diamond mode and long diamond mode – for various blowing ratios. For the CFD simulations, the non-conformal mesh interface, resulting due to the hexahedral elements on the primary domain and the tetrahedral elements on the coolant domain, was handled in ANSYS FLUENT. Velocity-inlet type boundary conditions were chosen by the authors to model the inflow conditions. The flow outlet was modeled using ‘pressure-outlet’ boundary conditions. The near wall boundary layer effected were resolved by the refined grids, characterized by wall Y+ in a range of 0 – 5 in their simulations. Based on their findings, the authors recommend the long-diamond mode of film cooling hole arrangement.

Much of the research work on film cooling had been focused in stationary conditions i.e. the specimen had been held stationary. Nabel Al-Zurfi [8] had investigated the film cooling methods for the gas turbine blades with the consideration of rotational velocity (0 rpm, 125 rpm, 250 rpm, 500 rpm). They had used simple-angled cylindrical holes on the suction side of the blades while the compound angled cylindrical holes on the pressure side. The Reynolds number corresponding to the flow conditions was 3 X 105. The authors had used the STAR CCM+ software for the CFD simulations. For the simulations, the inflow turbulence was specified using the turbulence length scale of one hole diameter. Their result indicated that the film cooling effectiveness increases with the rate of rotation. The authors had selected the Standard Smagorinsk-Lilly model to account for the flow turbulence. Interestingly, they had also compared the results from LES and RANS approach in terms of exit velocity ratio and the film cooling effectiveness. They concluded that the LES model predictions were in closed agreement with the experimental data as compared to the RANS models.

The turbulent models in RANS approach for the CFD simulations mostly use the iso-tropic flow turbulence. Jianqin Zhu [9] had proposed an anisotropic k-omega turbulence model, by introducing the rate of turbulence anisotropy. This was expected to prevent any under-prediction lateral vortex spreading. By introducing the anisotropy, the eddy viscosity in this model became a tensor from scalar. They had presented the results in terms of lateral averaged film cooling effectiveness at various Z/D sections. The anisotropy k-omega model provided superior performance compared to the iso-tropic k-omega and k-epsilon turbulence models.

Luca Andrei [10] had developed a decoupled approach to obtain the film cooling characteristics. A decoupled approach requires flow characteristics such as Temperature, Pressure, heat transfer coefficient, heat conduction through the solid and the internal cooling system of the blades. These were calculated by the authors using various solution methods such as CFD. They had studied the blade cooling for internally cooled turbine vane using NASA C3X:1983 blades and externally cooled turbine vane using NASA C3X:1988 blade profile. The necessary CFD simulations in the De-coupled approach were obtained by the authors using ANSYS CFX. They had employed the SST K-Omega turbulence model in their simulations.

The application film cooling for combustors of gas turbine had been investigated using transient CFD methods in ANSYS FLUENT by EhsanKianpour [11]. Film cooling holes of cylindrical and trenched shaped for the combustor end-wall with a blowing ratio of 3.18 was considered by the authors, for two hole alignment angle of 0° and 90°. The flow entry to the simulation domain was modeled using 'mass flow inlet' in their study. They had selected the RNG k-epsilon turbulence model for the simulations. The flow conditions in this research correspond to Re = 2.2 X 105. The authors suggested that the trenched hole film cooling methods nearly doubles heat transfer rate enhancement at the combustor end-wall surfaces.

The conjugate calculation techniques, that include the conduction and convection heat transfer, had been employed by many researchers to study the film cooling mechanisms. In the three-dimensional study conducted using CHTflow software, Norbert Moritz [12] had applied the conjugate calculation techniques for the gas turbine blade's leading edge film cooling. Their CFD solver was of finite volume based implicit formulation with multi-block mesh techniques. The closure of RANS in their CFD solver was provided by Baldwin-Lomax algebraic eddy-viscosity turbulence model. During the simulation, Fourier heat conduction equation was solved in the Solid Blocks and compressible, three-dimensional Navier-Stokes Equation was solved on the Fluid Blocks. The necessary coupling between the Fluid and Solid Blocks were imposed by the common wall temperature, arising from the equality of the local heat fluxes. The results had been discussed by the authors in the form of color plots for film cooling effectiveness. They had also compared the effectiveness on the blade leading edge from the experimental and the CFD simulations. The results were in good agreement, indicating the reliability of CFD simulations.

3. CONCLUSION

As per the recent study, it is needed to improve the efficiency of GTE, so the most common method is to increase the temperature at turbine inlet, which adversely affects turbine blade so to overcome this a new techniques is proposed. Most common blade cooling technique is the film cooling with single or multiple hole provided on the surface of the blade. In the existing methods the turbine blade cooling is always initiated by leading edge of the turbine blade. But if the cooling is proposed from trailing edge section of blade tip then can perform better

REFERENCES

[1] S. Friedrichs, H. P. Hodson and W. N. Dawes, "Distribution of Film Cooling Effectiveness on a Turbine Endwall Measured Using the Ammonia and Diazo Technique" International Gas Turbine and Aeroengine Congress & Exposition, Houston Texas; June 1995.

[2] Daniel G Hyams and James H Leylek, "A Detailed Analysis of Film Cooling Physics Part III: Streamwise Injection with Shaped Holes" International Gas Turbine and Aeroengine Congress & Exhibition, Florida; June 1997.

[3] Srinath V Ekkad, Je-Chin Han and Hui Du, "Detailed Film Cooling Measurements On a Cylindrical Leading Edge Model: Effect of Free-stream Turbulence and Coolant Density"International Gas Turbine and Aeroengine Congress & Exhibition, Florida; June 1997.

[4] K. T. McGovern, J. H. Leylek, "A Detailed Analysis of Film Cooling Physics: Part II – Compound – Angle Injection with Cylindrical Holes" Journal of Turbo machinery; vol.122, January 2000.

[5] J. E. Sargison, S. M. Guo, M. L. G. Oldfield, G. D. Lock, A. J. Rawlinson, "A Converging Slot-Hole Film-Cooling Geometry – Part 1: Low Speed Flat-Plate Heat Transfer and Loss" Journal of Turbomachinery; vol.124, July 2002.

[6] ReazHasan, AginPuthuKKudi, "Numerical Study of Effusion Cooling on an Adiabatic Flat Plate" Propulsion and Power Research; 2013.

[7] Yang Chengfeng, Zhang Jingzhou, "Influence of Multi-hole Arrangement on Cooling Film Development" Chinese Journal of Aeronautics; 2012.

[8] Nabeel Al-Zurfi, Ali Turan, "LES of Rotational Effects on Film Cooling Effectiveness and Heat Transfer Coefficient in a Gas-Turbine Blade with One Row of Air Film Injection" International Journal of Thermal Sciences; 2016.

[9] Jianqin Zhu, Kai Wang, ChaoqunNie, "Computational Research on Film Cooling under Rotating Frame by an Anisotropic Model" Propulsion and Power Research; 2013.

[10] Luca Andrei, Antonio Andreini, Bruno Facchini, Lorenzo Winchler, "A Decoupled CHT Procedure: Application and Validation on a Gas Turbine Vane with Different Cooling Configurations" Energy Procedia; 2013.

[11] EhsanKianpour, Nor AzwadiCheSidik, "Computational Investigation of Film Cooling from Cylindrical and Row Trenched Cooling Holes near the Combustor End-wall" Case Studies in Thermal Engineering; 2014.

[12] Norbert Moritz, KarstenKusterer, Dieter Bohn, Takao Sugimoto, Ryozo Tanaka, Tomoki Taniguchi, "Conjugate Calculation of a Film-Cooled Blade for Improvement of the Leading Edge Cooling Configuration" Propulsion and Power Research; 2013.

[13] Dibbon K. Walters and James H. Leylek, "A Detailed Analysis of Film Cooling Physics Part I: Stream-wise Injection with Cylindrical Holes" International Gas Turbine &Aeroengine Congress & Exhibition; Florida, 1997.

[14] Dibbon K Walters and James H Leylek, "A Systematic Computational Methodology Applied To A Three-Dimensional Film Cooling Flowfield" International Gas Turbine and Aeroengine Congress & Exhibition; Birmingham UK, June 1996.

[15] Guangchao Li, Chaolin Wu, Wei Zhang, Zhihai Kou, DaweiPeng, "Effect of Cross-flow direction of Coolant on Film Cooling Effectiveness with One.

[16] Prashant B.Kuyate et al., International Journal of Engineering Technology and Computer Research (IJETCR) © 2016 IJETCR.