ISSN: 2348 - 2117

International Journal of Engineering Technology and Computer Research (IJETCR)
Available Online at www.ijetcr.org
Volume 4; Issue 2; March-April-2016; Page No. 29-36

A Review on Film Cooling; Methods, Procedures, Analysis

Prashant B.Kuyate[1], Prof.N.C.Ghuge[2], Prof.D.D.Palande[3], Prof.V.S.Daund[4]

[1]PG Student, Matoshri College of Engineering & Research Centre, Nashik

kuyateprashant@gmail.com

[2]Associate Professor, Matoshri College of Engineering & Research Centre, Nashik

nilghuge@gmail.com

[3]Associate Professor, Matoshri College of Engineering & Research Centre, Nashik

palande1@rediffmail.com

[4]*Assistant Professor, Matoshri College of Engineering & Research Centre, Nashik*

Abstract

With the dependence of gas turbine efficiency on the turbine inlet temperature, various cooling methods had been applied to maintain the acceptable turbine blade temperature. Film cooling techniques is one of the efficient and more complex mechanisms. In this review journal, a comprehensive report on film cooling methods, procedures, applications, experimental approach towards film cooling, CFD simulation methods for film cooling, result analysis from the latest research had been compiled. The literature review had focused on the CFD simulation practices by the recent researchers in order to obtain the comparable results with experimental data. A summary from all these researches had also been provided in the concluding section of this paper.

Key Words: Film Cooling, Gas Turbine Blade Cooling, CFD, Film Cooling Effectiveness, etc.

Introduction

In order to obtain higher power output from the gas turbine engines, the turbine inlet temperature had been increased in the recent engines. This high temperature could lead to overheating of the blades and may result in blade failure. This could lead to engine failure and catastrophic effects. In order to prevent such failures, the blade surfaces must be maintained to acceptable limits. So, the cooling of blade surfaces must be considered. There are multiple cooling techniques such as Convective cooling, impingement cooling etc. to maintain the surface temperature to an acceptable range. Film cooling is one such cooling process. In film cooling, the surface will be covered with a thin film of cool air that acts as a buffer between the high temperature gas and the surface. The cold air will be injected at high pressures through the holes in the surface, resulting in a thin cold film surrounding the hot surfaces.

The common terminologies that were used in film cooling research had been provided here for the reference. The adiabatic film cooling effectiveness η = , where the subscripts w, c and ∞ represent wall, coolant and free-stream

Literature Review:

Based on the surface concentration of a coolant such as Ammonia gas, **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.

Daniel 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, by **Srinath 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 parameter in enhancing heat transfer rate.

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 **ReazHasan**[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 10^5. 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 10^5. 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.

Dibbon K Walters [13] had applied computational methods for jet-in-a-cross flow situation to study the relevant flow physics of film cooling. The CFD simulation was carried out using an implicit, pressure-correction solver with multi-grid. The authors had studied for two different film-hole length-to-diameter ratios and the flow conditions corresponding to three blowing ratios ranging from 0.5 to 2.0. From their studies, they had observed that the vorticity in the boundary layers within the film hole were primarily responsible for the secondary motion. Also, the authors noted that the selection of two-layer wall treatment instead of the Wall function approach lead to better simulation accuracy.

In another research, **Dibbon K Walters** [14], had simulated the discrete jet-in-cross-flow problem in film cooling using Fully Explicit, Time-marching, Reynolds Averaged Navier-Stokes, finite volume based solver. The major objective of this research work was to identify the sources of errors that make the CFD results to deviate from the experimental data. The test case that was considered by the authors was of 3-D Stream-wise Injection Jet, with a 35° injection angle. Their simulations were carried out using Standard k-epsilon turbulence model with wall functions. However, the authors had recommended that the near-wall modeling might provide accurate results over the Wall function approach. They also suggest the selection of turbulence model with the capability to include the anisotropy turbulence effects for more accurate result predictions from the CFD simulations.

The film cooling effectiveness shall be enhanced by injecting the coolant fluid in compound angles, resulting in a cross-flow of coolant-stream. The injected coolants from the two adjacent film holes interact and modify the vortex structure near the film cooling hole. **Guangchao Li** [15] had investigated the coolant cross-flow direction over the film cooling effectiveness. Their studies were conducted using the ANSYS FLUENT CFD solver. The authors had used the two-equation Realizable k-epsilon turbulence model. The second-order upwind difference scheme was chosen for interpolating the advection and diffusion at the cell faces. The authors had investigated the cross-flow impact on the flow and film cooling effectiveness. Based on their findings, they concluded that the coolant cross-flow directions significantly affect the flow profile.

By using CFX, **Younggi Moon** [16], had investigated the impact of the angle variation [0 to 75 degrees] between the primary and auxiliary film cooling holes of an anti-vortex hole. The film cooling effectiveness for varying blowing ratio [0.25 to 2.0] along with the free-stream turbulence were considered in this study. For the CFD simulations, the RNG k-epsilon turbulence model was chosen by the authors. The flow inlet to the computational domain was modeled using the 'velocity-inlet' boundary conditions while the 'pressure-outlet' boundary condition was applied for modeling the flow outlet. The authors had also conducted grid independence study by performing the simulations with 655,000, 1,246,000, 2,344,000 and 3,455,000 cells. For the final simulations, the 2,344,000 mesh was considered. Based on the results, the authors concluded that the film cooling effectiveness was strongly affected by the angle between the primary and auxiliary cooling holes.

Jin Wang [17] had conducted the experimental studies on a scaled down model (1/3rd model of GE-E3 turbine blade) to study the flow profiles of tip flow with 5 film cooling holes on the turbine blade. The influence on the flow profile by the rim width [0.9%, 2.1% and 3.0% of axial chord], and the groove depth [2.8%, 4.8% and 10% of blade span] were studied by the authors. The velocity field in the specimen was captured by the CCD camera while the pressure taps connected with the pressure gauges were employed to estimate the pressure profile. During the experimental studies, two semi-profiled blades were used to re-produce the periodic conditions. The inlet velocity for the experiment was varied from 1.5 m/s to 30 m/s and was measured using a pitot tube.

Yuting Jiang [18] had employed water mist as coolant for enhancing the film cooling effectiveness. The Eulerian-Lagrangian particle tracking method in the CFD simulations were applied in the simulations by the authors. In this two-phase cooling method, the influence of critical parameters such as mist concentration [1%, 3% and 5%], mist diameter [5, 10 and 15 microns], the location of mist injection [leading edge, pressure side and suction side] over the film cooling enhancement was studied by the authors. For the particle injection, the Rosin-Rammler and Uniform droplets distribution was considered in the work. The SST K-Omega turbulence model was used in these simulations. The results were discussed in the graphical form of heat transfer coefficients plotted along the span direction. From their research, the authors had identified the wall temperature reduction of 20 to 48 K for the mist concentration of 1-3%. Also, the uniform droplets with smaller size offer rapid and effective evaporation and droplet dynamics that resulted in better heat transfer enhancements

K. Thole[19] had, in their research work, explained the procedure to measure flowfields surrounding the film cooling holes. They had conducted the experiments with three different hole geometries such as round hole, a hole with laterally expanded exits and a hole with a forward-laterally expanded exit. The authors had used a two component, coincident, fiber optic Laser Doppler Velocimeter (LDV) for measuring the velocity fields. In order to measure the near-wall measurements, the LDV was rotated at 45 degrees and tilted at 4.5 degrees. Oil Particles were used to seed the flow path in the primary and the coolant passage. The lateral spread of the cooling jet was studied by measuring velocity profiles in vertical direction at 14 stream-wise locations.

NASA C3X, a very well researched turbine vane, having film cooling arrangements – three rows at the leading edge and two rows each on pressure and suction sides- was investigated for pulsed film cooling using CFD methods by **ZhaoqingKe**[20]. The pulsed film cooling was generated by square and sinusoidal waves by the authors for three blowing ratios [0.5, 0.75 and 1.0]. With the pulsed film cooling, the amount of coolant in circulation will be less as compared to the steady operation wherein the continuous supply of coolant flow is required. In these CFD simulations, the authors had employed tetrahedrons mesh with 30 layers of prism mesh that resulted in wall Y+ in the range of 2. ANSYS CFX was used for the simulations with the SST k-omega turbulence model and High Resolution Advections schemes for discretization. The authors had observed from their results that the film cooling effectiveness reduces at lower blowing ratio and increases with the blowing ratio. The sinusoidal waves pulsed film cooling had high effectiveness as compared to the steady operations.

The flow filed observed in the vicinity of the film cooled turbine vanes were of complex and anisotropic turbulence in nature. Much of research had been focused in modeling iso-tropic turbulence, there had been a need to accurately model the Reynolds stresses with the help of anisotropic formulation. **Xueying Li** [21] had developed an algebraic anisotropic turbulence model. In this model, the anisotropy in the Reynolds stresses had been introduced by including the anisotropic eddy viscosity ratio. The inclusions of anisotropic corrections to the eddy viscosity had a direct impact on momentum equation that influences the scalar transport flux on convective heat transfer. The authors had compared these results with the RANS models as well as against the experimental data and observed to have better co-relation for the experimental data in comparison to RANS models.

The injection location of coolant in the film cooling arrangement influences the secondary vortices creation which impacts the film cooling effectiveness. **Xueying Li** [22] had investigated the film cooling for the distribution of injection hole for a turbine vane. The authors had conducted the experiments along with the numerical simulations. The CFD simulations were performed using the boundary conditions such as ‘mass-flow-inlet’, ‘pressure-outlet’ and ‘wall’ with the relevant flow conditions. Considering the low-speed conditions, the incompressible solution approach had been considered with SIMPLE algorithm proving the linkage between the pressure and the velocity. The authors argue that the local blowing ratio of the end-wall film cooling increased with the local static pressure. Since the cooling jets at the high local blowing ratio tend to be sensitive to the supply pressure, the film cooling holes located near the high static pressure locations could achieve wide range of blowing ratios.

Xueying Li [23]had varied conducted experiment as well as numerical simulations to study the influence of the film cooling effectiveness by varying the number of cooling jet rows. They had considered single row film cooling, three combinations of two-row film cooling and full coverage configuration with five row cooling arrangements in their study. The CFD simulation for this work followed the earlier work by the authors [22].The experiments were conducted using PSP technique by using the light emitting characteristics of PSP while at excitation. This light emission is inversely proportional to the oxygen pressure.

An investigation on the inclusion of coolant flow swirl had been studied by **Zhao Liu** [24] using numerical simulations with the help of ANSYS CFX. They had studied for two rectangular shaped inlet sections for the coolant injections. They had also varied the jet nozzle’s aspect ratio, coolant jet angle [30, 60, 90 and

120 degrees] along with the flow conditions defined by Reynolds Number [5,414 to 10,000]. The inlet total temperature in their study was 239.85 K. Air was modeled as ideal-gas with the inlet turbulence intensity was defined as 1%. The authors had performed the CFD simulations using four turbulence models – Standard K-Epsilon, RNG K-Epsilon, Standard K-Omega and SST K-Omega – and had observed that SST k-Omega provided better co-relation with experimental data.

The film cooling with branched coolant jet injection was studied by **Yu Yao** [25] with the help of numerical simulations in ANSYS FLUENT for four blowing ratios [0.5, 1.0, 1.5 and 2.0]. The flow conditions considered by the authors include the inlet temperature of 600 K with the inlet velocity of 30 m/s. The coolant jet was injected at a temperature of 300 K. The inlet turbulence was specified using 5% turbulence intensity. The authors had applied symmetry boundary conditions in the side walls. The thermal conditions on the walls plate surface was modeled using the 'heat-flux' wall boundary conditions in the simulations. For these simulations, the RNG k-epsilon turbulence model was considered by the authors. The adiabatic film cooling effectiveness at various locations had been discussed in a graphical form. It had been observed that the film cooling effectiveness increases with the increase in blowing ratio. They had also noted that the increased film cooling effectiveness in the branched coolant jet injection was achieved with relatively higher flow losses.

The coolant jet' compound angle and its influence on film cooling effectiveness was studied numerically by **Yanmin Qin** [26]. The effectiveness of the coolant injection will be critically based on the surface curvature and the stream-wise pressure gradient. The authors had used the PSP – Pressure Sensitive Paint – technique in their experimental studies. Based on their findings, the authors noted that the film cooling effectiveness increases due to the compound angle only for the high blowing ratio. The enhancement weren't significant at lower blowing ratio.

With the help of Liquid Crystal Thermography, **Francesca Satta**[27] had investigated the impact of discrete-hole arrangement over the film cooling effectiveness of the turbine blade cascade' end-wall. The coolant holes – diameters of 5 mm – were arranged in two configurations in this research work. Their experiments had been conducted with cold air as primary fluid and the hot air as secondary fluid with the temperature difference ranging from 9 -35 K. The authors had observed that the standard arrangement of cooling holes resulted over-cooled regions near the injection holes while the region between the coolant holes were under-cooled. This had been overcome by modifying the coolant hole arrangement.

Zhu Xingdan[28] had conducted experiment studies using Infrared Radiation (IR) Thermography techniques to study the film cooling on a turbine vane. The authors had varied the cooling jet angle from 35 to 90 degrees. The film cooling holes were of 2 mm diameter with the pitch-to-diameter ratio of 2.55. In this research work, they had varied the blowing ratio 0.44 to 2.64 by controlling the coolant flow rate. They had considered the flow conditions corresponding to Re = 102,853. From their results of adiabatic film cooling effectiveness, they had observed that the adiabatic film cooling effectiveness increases along the blade chord from the leading edge stagnation and starts decreasing. This could be attributed to the intense mixing of coolant and the free-stream gas near the leading edge which result in high film cooling effectiveness.

In their research work, **Tan Xiao-ming**[29] had investigated the effectiveness of impingement cooling methods with film cooling holes angled normal to the mainstream flow. They had used polycarbonate test plate as well as stainless steel plate for their experiments. Their research work focused on blowing ratio [varied from 0.6 to 2.4], hole arrangement [in-line and staggered], hole to hole pitch ratio [varied from 3 to 5] and jet to target spacing ratio [varied from 2 to 4] over the film cooling effectiveness. The authors had observed almost 10 – 15% increase in the film cooling effectiveness in the staggered arrangement of cooling holes over the in-line arrangements. Also, reducing the jet-to-target ratio from 4 to 2 enhances the cooling effectiveness.

Xing Yang [30]had conducted numerical simulations using ANSYS CFX film cooling with swirling motion of coolant. They had used the cylindrical, clover shaped and compound angle holes in their studies. The authors had applied the Reynolds Averaged Navier Stokes (RANS) approach in their simulations with SST k-omega turbulence model being employed. The blowing ratio for this study was in the range of 0.5 to 1.5. The swirling motion of coolant flow had strong influence over the film cooling effectiveness, even more so at the higher blowing ratio.

Observations:

It's recommended to perform the grid independence study while conducting CFD simulations. Reynolds Averaged Navier Stokes (RANS) based turbulence models in the numerical approach produces comparable results against experimental data.

In the CFD simulations for film cooling, symmetry boundary conditions shall be applied in order to limit the number of grid cells and subsequently the computational time, without compromising the solution accuracy.

Pressure Sensitive Paints (PSP) that works based on light emission based on Oxygen pressure could be used for film cooling experimental methods.The adiabatic film cooling effectiveness, a non-dimensional parameter, will be higher near the leading edge and decreases along the blade chord length.

Film cooling hole arrangement – staggered or in-line – have strong influence over the film cooling effectiveness. Also, the film cooling hole shape plays a critical role in the enhancement of heat transfer rate and the resulting film cooling effectiveness.
In case of high blowing ratios, pulse film cooling technique could be considered for enhancing the film cooling effectiveness.

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. 2Guangchao Li, Chaolin Wu, Wei Zhang, Zhihai Kou, DaweiPeng, "*Effect of Cross-flow direction of Coolant on Film Cooling Effectiveness with One*

Inlet and Double Outlet Hole Injection” propulsion and Power Research; 2012

16. Younggi Moon, Soon Sang Park, Jung Shin Park, Jae Su Kwak, “*Effect of Angle between the Primary and Auxilary Holes of an Anti-Vortex Film Cooling hole*” 2014 Asia-Pacific International Symposium on Aerospace Technology

17. Jin Wang, BengtSunden, Min Zeng, Qiuwang Wang, “*Film Cooling Effects on the tip flow charecteristics of a gas turbine*” Propulsion and Power Research; 2015; 4(1)

18. Yuting Jiang, QunZheng, Ping Dong, Jianhui Yao, Hai Zhang, JieGao “*Conjugate Heat Transfer Analysis of Leading Edge and Downstream mist-air film cooling on turbine vane*” International Journal of Heat and Mass Transfer, 2015;

19. K.Thole, M. Gritsch, A. Schulz and S. Wittig, “*Flowfield Measurements for Film Cooling Holes with Expanded Exits*” International Gas Turbine and Aeroengine Congress and Exhibition; June 1996.

20. ZhaoqingKe, Jianhua Wang, “*Numerical Investigations of Pulsed film Cooling on an entire Turbine Vane*” Applied Thermal Engineering, 2015;

21. Xueying Li, Jing Ren, Hongde Jiang, “*Application of Algebraic Anisotropic Turbulence models to film cooling*” International Journal of Heat and Mass Transfer, 2015;

22. Xueying Li, Jing Ren, Hongde Jiang, “*Film Cooling Effectiveness Distribution of Cylindrical Hole Injections at Different Locations on a Vane Endwall*” International Journal of Heat and Mass Transfer, 2015;

23. Xueying Li, Jing Ren, Hongde Jiang, “*Multi-row Film Cooling Characteristics on a Vane End-wall*” International Journal of Heat and Mass Transfer, 2015;

24. Zhao Liu, Jun Li, ZhenpingFeng, Terrance Simon, “*Numerical Study on the effect of Jet Nozzle Aspect Ratio and Jet Angle on Swirl Cooling in a Model of a Turbine Blade Leading Edge Cooling Passage*” International Journal of Heat and Mass Transfer, 2015;

25. Yu Yao, Jingzhou Zhang, Yang Yang, “*Numerical Study on Film Cooling Mechanism and Characteristics of Cylindrical Holes with branched Jet Injections*” Propulsion and Power Research, 2013;

26. Yanmin Qin, Xueying Li, Jing Ren, Hongde Jiang, “*Effects of Compound Angle on Film Cooling Effectiveness with different Stream-wise Pressure Gradient and Convex Curvature*” International Journal of Heat and Mass Transfer, 2015;

27. Francesca Satta, Giovanni Tanda, “*Effect of Discrete-hole Arrangement on Film Cooling Effectiveness for the End-wall of a Turbine Blade Cascade*” Applied Thermal Engineering, 2015;

28. Zhu Xingdan, Zhang Jingzhou, Tan Xiaoming, “*An Experimental Investigation of Showerhead Film Cooling Performance on a Turbine Blade*” Asia Pacific International Symposium on Aerospace Technology, 2014;

29. Tan Xiao-ming, Zhang Jing-zhou, XuHua-sheng, “*Experimental Investigation on Impingement / Effusion Cooling with Short Injection Holes*” International Communications in Heat and Mass Transfer, 2015;

30. Xing Yang, Zhao Liu, Zhansheng Liu, ZhenpingFeng, “*Numerical Analysis on Effects of Coolant Swirling Motion on Film Cooling Performance*” International Journal of Heat and Mass Transfer, 2015.