Investigation of Film Cooling Technique for Cylindrical Leading Edge Model using CFD and Experimental Method

#1 Prashant B. Kuyate, #2 Prof. N. C. Ghuge

[1]Department of Mechanical Engineering, PG Student, Savitribai Phule Pune University, MCOERC Nashik, India
[2]Department of Mechanical Engineering ,Associate Professor, Savitribai Phule Pune University, MCOERC Nashik, India
[3]Department of Mechanical Engineering ,Assistant Professor, Savitribai Phule Pune University, MCOERC Nashik, India

Abstract: Film cooling technique is one of the efficient mechanisms applied on gas turbine blades. This is investigated experimentally and with CFD approach in this research work. The model of cylindrical shape with the aspect ratio of 2 is considered for this study. The study was conducted with the film cooling holes arranged in different angles 15, 30 and 45 degrees from the stagnation. The CFD simulation was conducted using ANSYS FLUENT. Flow conditions were varied from Re = 9000 to Re = 25,000 in this investigation. The RNG k-epsilon turbulence model was chosen for these simulations. It is observed that the flow distribution surrounding the cylinder is better for the 15 deg cooling hole arrangement. This results in better performance in terms of high heat transfer coefficient as well as low surface temperature. The results predicted from the experimental studies are also having the same pattern.

Keywords: Film Cooling, Gas Turbine Blade Cooling, CFD, Film Cooling Effectiveness.

1. 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 objectives for the project are:

1. To investigate the impact of locations of film cooling holes on Film Cooling Effectiveness for various velocities (for θ = 15, 30 and 45 degrees)
2. To investigate the impact of locations of film cooling holes on Film Cooling Effectiveness for various temperature differences (ΔT = 40, 50, 60, 70 K)

These configurations will be studied for the varying inflow conditions, characterized by the Reynolds number corresponding to 8,215 to 32,860. In order to compare the enhancement in heat transfer rate, due to the film cooling, a separate study will be conducted without the film cooling arrangement for the identical flow and thermal conditions and the results will be compared. Based on the surface concentration of a coolant such as Ammonia gas, (S. Friedrichs,et al, 1995) 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. The critical film cooling parameters like the cold/hot fluid density ratio, and the blowing ratio were studied by (Daniel G Hyams ,et al,1997) using steady-state, 3-D simulations using RANS approach. Their study included the experimental as well. For a cylindrical leading edge, the film cooling effectiveness and the resulting heat transfer coefficient was studied experimentally for the flow conditions of Re = 100,900, by (Srinath V Ekkad,et al,1997) with the coolant injection holes arranged in rows at ± 15° from the stagnation point. Their study focused on obtaining the film cooling effectiveness using liquid crystal technique.(K. T. McGovern and J. H. Leylek,2000) 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. The film cooling heat transfer rate and the associated aerodynamic losses were experimentally studied by (J. E. Sargison,et al,2002). Another research by (Reaz Hasan,Agin Puthukkudi,2013) 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. 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 and Zhang Jingzhou,2012). 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. 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 and Ali Turan,2016) 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). The authors had used the STAR CCM+ software for the CFD simulations. The turbulent models in RANS approach for the CFD simulations mostly use the iso-tropic flow turbulence. (Jianqin Zhu,et al,2013) had proposed an anisotropic k-omega turbulence model, by introducing the rate of turbulence anisotropy. The anisotropy k-omega model provided superior performance compared to the iso-tropic k-omega and k-epsilon turbulence models.

(Luca Andrei,et al,2013) had developed a decoupled approach to obtain the film cooling characteristics. 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 application film cooling for combustors of gas turbine had been investigated using transient CFD methods in ANSYS FLUENT by (Ehsan Kianpour and Nor Azwadi Che Sidik,2014).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°. In the three- dimensional study conducted using CHT flow software, (Norbert Moritz,et al,2013) had applied the conjugate calculation techniques for the gas turbine blade's leading edge film cooling. (Dibbon K Walters and James H Leylek,1996) had applied computational methods for jet-in-a-cross flow situation to study the relevant flow physics of film cooling. In another research, (Dibbon K Walters and James H Leylek,1996) 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. (Guangchao Li,et al,2012) had investigated the coolant cross-flow direction over the film cooling effectiveness. By using CFX, (Younggi Moon,et al,2014),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 was considered in this study. (Jin Wang,et al,2015) 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. (Yuting Jiang ,et al,2015) 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. (K. Thole, et al,1996) had, in their research work, explained the procedure to measure flowfields surrounding the film cooling holes. 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 (Zhaoqing Ke and Jiahua Wang,2015). 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.

Nomenclature

Sr. No.	Nomenclature	Description
1)	Re	Reynolds Number
2)	Nu	Nusselt Number
3)	V	Flow Velocity, m/s
4)	T	Temperature, K
5)	H	Convection heat transfer Coefficient, W/m2-K
6)	K	Thermal conductivity, W/m-K
7)	D	Diameter
8)	ΔT	Change in temperature

9)	HTC	Heat Transfer Coefficient
10)	dP	Change in pressure
11)	Q	Heat transfer
12)	θ	Hole inclination angle
13)	PG	Pressure Gauge
14)	TS	Temperature Sensor
15)	CV	Control Valve

2. Experimentation

The film cooling mechanism applied on a cylindrical mechanism is investigated for the Influence of hole arrangement (15, 30 and 45 degrees from the stagnation) over the cooling effectiveness.

A) Impact of inflow temperature differences (ΔT = 40, 50, 60 and 70).

B) Varying flow conditions – characterized by Reynolds number (9780 ≤ Re ≤ 25,000). The study was conducted using Experimental and CFD methods.

Table 1. Flow Conditions

Reynolds Number	Velocity, m/s
9780	1.0
14,670	1.5
19,560	2.0
24,450	2.5

The study was conducted for various operating conditions to develop the complete understanding of the cooling mechanism. The flow conditions ranging from Re = 9,000 to Re = 25,000 was considered. Thermal conditions for this study were defined based on the difference between the hot inlet temperature and the cold air inlet temperature (ΔT).

Table 2.variation of Temperature Differences

Cold Inlet Temp., K	Hot Inlet Temp.,K	Temp.Difference, ΔT, K
298	338	40
298	348	50
298	358	60
298	368	70

The cylindrical specimen was placed in its location and the blower operation started. At first, the blower is turned on to supply the air. From the anemometer readings, or from orifice meter the stability of the flow is monitored. Once the flow stability is achieved, the electric heater will be turned on to supply the thermal energy to the flow. Now, the temperature from TS-01 and TS-03 is to be monitored for steady state conditions. During this, the control valve CV-01 is closed thus preventing the cold air in to the cylindrical specimen.

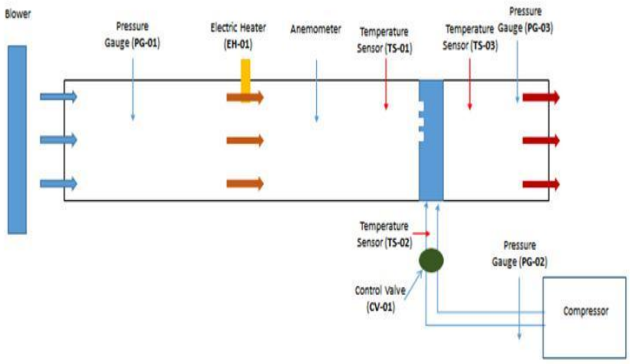

Fig.1: Schematic of Experimental Set-up

Fig.2: Experimental Setup

After the temperature readings are stabilized, an indication of steady state conditions, the control valve CV-01 is to be opened for the cold air to pass through and form the cold air thin film surrounding the cylindrical surfaces.

Again, the temperature readings from TS-01, TS-02 and TS-03 are monitored and the steady state values are noted. The same steps will be repeated for the successive configurations.

2.1 Geometry of specimen

The specimen had three rows of film cooling holes.

- No of holes : 19
- Hole diameter: 3 mm placed at uniform interval
- Cylinder Height: 100 mm
- Cylinder Inner Diameter: 30 mm
- Cylinder Outer Diameter: 50 mm
- The film cooling holes were made at 15°, 30° and 45°
- The angles were in reference from the stagnation point

2.2 Calculations

Reynolds Number,

$$Re = \frac{\rho v D}{\mu} \quad (1)$$

Nusselt Number,

$$Nu = \frac{h.Lc}{K} \quad (2)$$

The adiabatic film cooling effectiveness,

$$\eta = \frac{T_w - T_\infty}{T_c - T_\infty} \quad (3)$$

Where, the subscripts w, c and ∞ represent wall, coolant and free-stream

3. CFD Methodology

3-Dimensional, Steady-state CFD simulations were performed using ANSYS FLUENT V16.0. This project work includes the convection heat transfer from the solid surfaces to the adjacent air; and the conduction heat transfer in the cylindrical specimen. This combined mode heat transfer is called 'conjugate heat transfer'. The 'pressure based solver' was chosen for the simulations with the consideration to the low speed flows in this problem. Air density and other properties such as viscosity, specific heat will be considered as constant values. Based on the literature survey, the following appropriate boundary conditions were imposed on the model. Fluid inlet: Velocity inlet boundary condition [velocity, static temperature]. Fluid Outlet: Pressure Outlet [static pressure]. Walls: Adiabatic, No-slip, stationary walls.

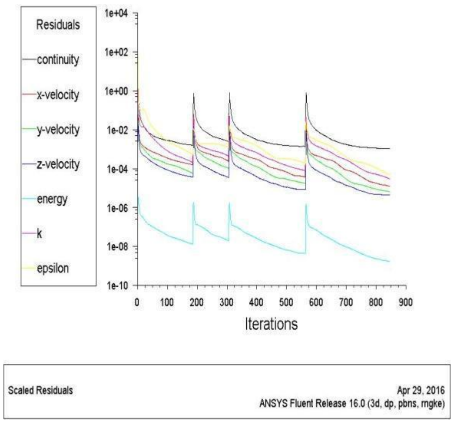

Fig.3: Scaled Residuals

4. Results and Discussion

The coolant flow leaves through the film cooling holes as can be seen in the 'Side Section View' as well as 'Top Section View'. This low temperature fluid forms the film around the 'cylindrical specimen' which resulted in lower surface temperature.

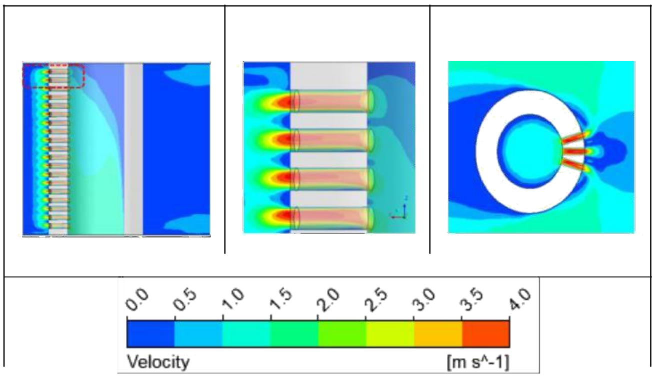

Fig.4: Velocity Contours for 15 degree Film Cooling Holes

In the following temperature contours, near the film cooling injection location, the low temperature region caused due to the injection of coolant jet could be observed.

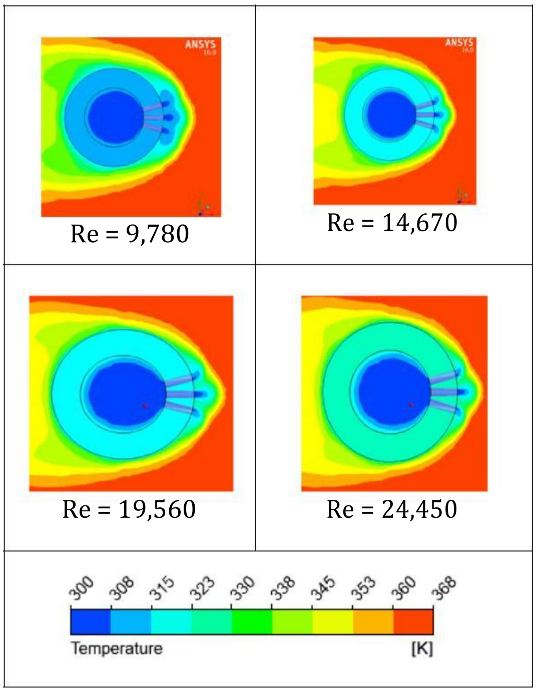

Fig.5: Temperature Contours

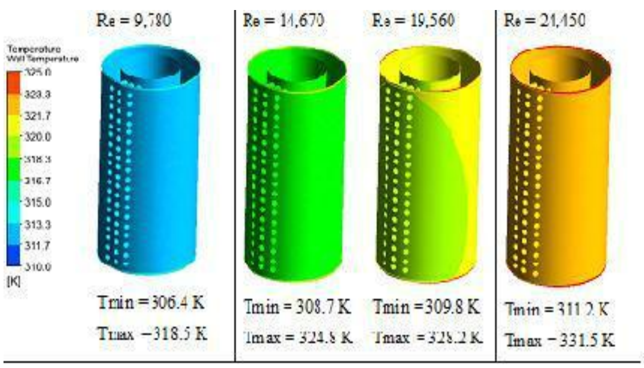

Fig.6: Surface Temperature Contours for 15 degree Cooling Hole Arrangement

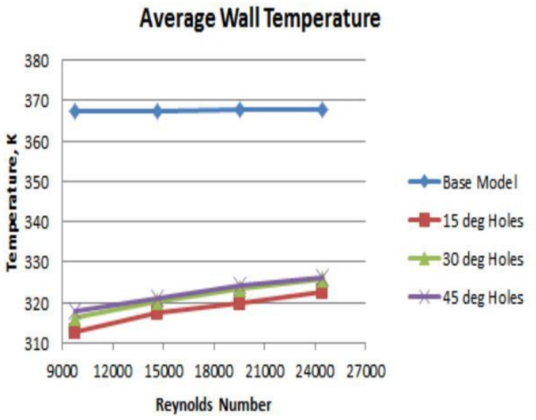

Fig.7: Comparison of Average Wall Temperature for all Configurations

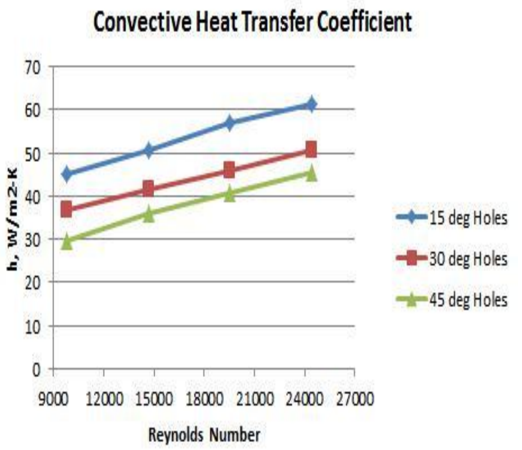

Fig.8 : Comparison of heat transfer coefficient for all Configurations

The experimental studies for the 15° film cooling hole configuration was conducted also extrapolation was done using statistical techniques. The temperature readings from the thermocouple were averaged and were compared against the CFD predictions. The data are provided in the following table.

Average Wall Temperature, K			
Re	Experiment	CFD	% Difference
9780	329	312.9	4.89 %
14670	337	317.5	5.78 %
19560	345	319.9	7.27 %
24450	352	322.8	8.29 %

Table 3: Comparison of experimental & CFD results

5. Conclusions

1) The averaged surface temperature of the cylinder reaches lower value for the 15 degree cooling hole arrangement.
2) This indicates the better cooling arrangement as compared to the other configurations. Heat transfer coefficients are also good than other configurations.
3) Also, as compared to the base model, the surface temperature reduction was in the range of 60 K, which could be considered significant.
4) The deviation of CFD results from the experimental data was with-in acceptable limits and serves as a validation for this research work.

References

S. Friedrichs,H. P. Hodson and W. N. Dawes,(1995) "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*.

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

Srinath V Ekkad, Je-Chin Han and Hui Du,(1997), "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*.

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

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

Reaz Hasan, Agin PuthuKKudi,(2013) "Numerical Study of Effusion Cooling on an Adiabatic Flat Plate" *Propulsion and Power Research.*

Yang Chengfeng, Zhang Jingzhou,(2012),"Influence of Multi-hole Arrangement on Cooling Film Development" *Chinese Journal of Aeronautics*.

Nabeel Al-Zurfi, Ali Turan,(2016), "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*.

Jianqin Zhu, Kai Wang, Chaoqun Nie,(2013), "Computational Research on Film Cooling under Rotating Frame by an Anisotropic Model" *Propulsion and Power Research.*

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

Ehsan Kianpour, Nor Azwadi Che Sidik,(2014), "Computational Investigation of Film Cooling from Cylindrical and Row Trenched Cooling Holes near the Combustor End-wall" *Case Studies in Thermal Engineering.*

Norbert Moritz, Karsten Kusterer, Dieter Bohn, Takao Sugimoto, Ryozo Tanaka, Tomoki Taniguchi,(2013), "Conjugate Calculation of a Film- Cooled Blade for Improvement of the Leading Edge Cooling Configuration" *Propulsion and Power Research.*

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

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

Guangchao Li, Chaolin Wu, Wei Zhang, Zhihai Kou, Dawei Peng,(2012)"Effect of Cross-flow direction of Coolant on Film Cooling Effectiveness with One Inlet and Double Outlet Hole Injection" *propulsion and Power Research.*

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

Jin Wang, Bengt Sunden, Min Zeng, Qiuwang Wang,(2015),"Film Cooling Effects on the tip flow charecteristics of a gas turbine" *Propulsion and Power Research.*

Yuting Jiang, Qun Zheng, Ping Dong, Jianhui Yao, Hai Zhang, Jie Gao,(2015),"Conjugate Heat Transfer Analysis of Leading Edge and Downstream mist -air film cooling on turbine vane" *International Journal of Heat and Mass Transfer.*

K.Thole, M. Gritsch, A. Schulz and S. Wittig,(1996), "Flowfield Measurements for Film Cooling Holes with Expanded Exits" *International Gas Turbine and Aeroengine Congress and Exhibition.*

Zhaoqing Ke, Jianhua Wang,(2015),"Numerical Investigations of Pulsed film Cooling on an entire Turbine Vane" *Applied Thermal Engineering*.