INTERNATIONAL JOURNAL OF MECHANICAL ENGINEERING AND TECHNOLOGY (IJMET)

ISSN 0976 – 6340 (Print)
ISSN 0976 – 6359 (Online)
Volume 5, Issue 9, September (2014), pp. 91-100
© IAEME: www.iaeme.com/IJMET.asp
Journal Impact Factor (2014): 7.5377 (Calculated by GISI)
www.jifactor.com

IJMET
©IAEME

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF ADIABATIC FILM COOLING EFFECTIVENESS OVER THE COMPOUND ANGLED GAS TURBINE BLADE LEADING EDGE MODEL

Giridhara Babu Y.[1*], Ashok Babu T.P.[2], Anbalagan M.[3], Meena R.[4]

[1]Propulsion Division, CSIR-National Aerospace Laboratories, Bangalore, India,
[2]Mechanical Engg. Dept., National Institute of Technology Karnataka, Mangalore, India
[3, 4]Mechanical Engg. Dept., The Oxford College of Engineering, Bangalore, India

ABSTRACT

This study aims at investigating the film cooling effectiveness using both experimentally and numerically for the scaled up gas turbine blade leading edge compound angle model. A compound angle gas turbine blade leading edge model having the five rows of holes, one at stagnation line, two rows of holes at 30 degrees on either side of stagnation line and two rows of holes at 60 degrees on either side of stagnation line. Each row has the five holes at a pitch of 21mm with the varied hole angles of 0, 30, 45, 55 and 60 degrees oriented with the stream line direction. The film cooling hole rows are arranged in a staggered manner to cover the more flow area on the blade surface and with each row consisting of 5 holes with the hole diameter of 4mm. Experiments are conducted by varying the blowing ratios (B.R) in the range of 1.0 to 2.5 at the density ratio (D.R) of 1.30, at a nominal flow Reynolds number of 1, 00,000 based on the leading edge diameter. The blowing ratio is varied by varying the coolant mass flow with the main flow maintained at a constant rate. The density ratio of 1.30 is maintained by allowing the coolant flow at 231K and main stream at room temperature. The film cooling effectiveness is also found using CFD (Fluent) simulation. The k-€ realizable turbulence model is used to solve the flow field. The generated CFD results are compared with the experimental results for the validation of CFD. The CFD results indicated the similar trends of the cooling effectiveness results as that of experimental values. Both the CFD and Experimental shown the increase in cooling effectiveness increases with the increase in blowing ratio upto 2.0 and found the decrease over the 2.0 for this model hence, the optimized blowing ratio can be considered as 2.0. The generated temperature and velocity contours using CFD shown the heat loads on the leading edge surface.

Keywords: Blowing ratio, CFD**,** Compound angle, Density ratio, Gas turbines.

1. INTRODUCTION

Generally, the Gas turbines are operated at higher temperatures in the range of 1200-1800 degree Celsius so the materials used for gas turbine blades should with stand these high temperatures without any melting and thermal stresses. The gas turbine blade leading edges are the vital parts in the turbines as they are directly hit by the hot gases, hence the optimized cooling of gas turbine blade leading edge surfaces is essential. Film cooling is used in many applications to reduce convective heat transfer to a surface. In order to increase the life of the blade and efficiency, the optimized cooling of gas turbine blade leading edge surfaces is essential. Film cooling is used in many applications to reduce convective heat transfer to a surface. Some of the examples are, film cooling of gas turbine combustion chamber liners, vanes and blades which are subjected to high heat loads from combustion gases.

Film cooling is the introduction of a secondary fluid at one or more discrete locations along a surface exposed to a high temperature environment to protect the surface not only in the immediate region of injection but also in the downstream region as shown in Fig.1. Turbine airfoil surfaces, shrouds, blades tips and end walls are cooled using discrete-hole film cooling. Film cooling protects the airfoil surface directly, compared to internal cooling technique that remove heat from the inside surface. Film cooling also removes heat from the blade surface through the film hole by internal convection, $q= h_o (T_f -T_w)$, and film cooling is defined as, $\eta = (T_m - T_f) / (T_m - T_c)$.

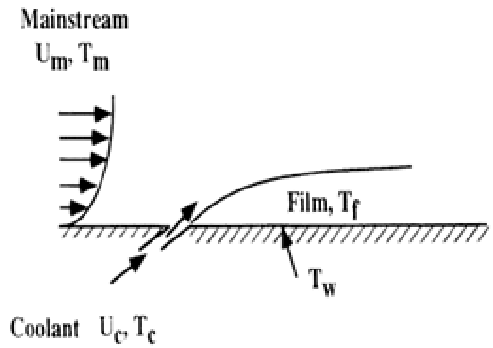

Fig. 1: Schematic of Film Cooling Concept

The thermal protection is expected to provide reduced heat load to the airfoil surface. Designers need to know the net heat load into the component surface when film is injected. Due to complex nature of discrete hole injection, there is need to know the local wall temperature (T_w) under the film and the gas side heat transfer coefficient with film injection. Both these components are required to estimate reduced heat load to the surface.

Film cooling primarily depends on the coolant-to hot mainstream pressure ratio (Pc/Pt), temperature ratio (Tc/Tg), and the film cooling hole location, configuration and distribution on a film cooled airfoil. The coolant-to-mainstream pressure ratio can be related to the coolant-to-mainstream mass flux ratio (blowing ratio) the coolant-to mainstream temperature ratio can be related to the coolant-to-mainstream density ratio. In a typical gas turbine airfoil, the blowing ratios vary approximately from 0.5 to 2.50 while the Tc/Tg values vary from 0.5 to 0.85, corresponding density ratios approximately in the range of 1.3 to 1.5.

Sang Woo Lee and Young Beom Kim [1], Surface flow visualizations show that the increase in the orientation angle results in better film coverage, especially in the spanwise direction, but produces more flow disturbances such as flow reversal and recirculation. A near-wall flow model for the velocity ratio of 2.0 has been proposed from the visualizations. the strength of downstream secondary flow strongly depends on the velocity ratio.

Srinath Ekkad and Chin Han JE [2], studied the recent development in turbine blade film cooling accurate and detailed local heat transfer and film cooling for turbine edge region would be useful to prevent blade failure due to local heat spot. Flow visualization/ measurement, and the CFD predictions would provide valuable information for designing effective cooled blades for advanced gas turbines.

D.G. Bogard and K. A. Thole [3] studied the gas turbine film cooling; the film-cooling performance is closely linked to whether the coolant jet has separated from the surface. For nominal conditions of a flat surface, low Freestream turbulence, and cylindrical holes, the film-cooling performance is reasonably predictable with empirical correlations. However, surface curvature, high Freestream turbulence, and shaping of the hole exit can greatly change film-cooling performance by significantly affecting the blowing ratio at which the coolant jet separates. CFD predictions, though very useful in providing insight in the spatial details of the film-cooling process, are also limited by the very complex flow conditions that occur for film cooling, particularly when the coolant jets begin to separate. Consequently, the film-cooling performance for actual turbine conditions is often difficult to predict precisely, and this remains a major constraint in the design for the durability of the turbine section.

This study aims at investigating the film cooling effectiveness using both experimentally and numerically for the scaled up gas turbine blade leading edge compound angle model. The film cooling hole with the hole diameter of 4mm. Experiments are conducted by varying the blowing ratios in the range of 1.0 to 2.5 at the density ratio of 1.30, at a nominal flow Reynolds number of 1, 00,000 based on the leading edge diameter.

2. EXPERIMENTAL SETUP AND PROCEDURE

2.1 Model Description

Only the semicircular leading edge portion of gas turbine blade is taken in to our study, the model is generated using solid works and fabricated using rapid prototyping, using the model description as shown in Table 1, ABS-M30 material is use to fabricate the leading edge model which is a low thermal conductivity material to avoid heat losses from the gas path non-contact side of the model.

TABLE 1: Fabricated Turbine Blade Leading Edge Model Description

SI.No.	Model Description	Dimensions
1	Leading Edge Outer Diameter	89 mm
2	Leading Edge Inner Diameter	65 mm
3	Film Cooling Hole Diameter and Pitch	D=4 mm and Pitch= 21mm
4	Leading Edge Model Height	210 mm
5	No. of Rows	5
6	Compound Hole angles	0, 30, 45, 55 and 60 Deg. with Stream line direction
7	Hole Orientation Angle	0, 30 and 60 Deg. Angles from stagnation line

The test models are prepared half cylindrical with flat downstream surfaces by attaching the coolant chamber. Hard foam is filled in the model slots to have the further low thermal conductivity. The Fig.2 and Fig.3 shows a fabricated model of turbine blade leading edge scaled up configuration. Stainless steel sheet having thickness of 0.15mm with a required film cooling hole geometry, machined by water jet cutting is wound over the model. The SS sheet with an area of 260 x 160mm is connected in series by brass bus bars to supply the high current at low voltage to heat the model. The reference thermocouples are soldered underside of the SS sheet for applying the correction factor to the thermo gram data and these are routed through the model slots.

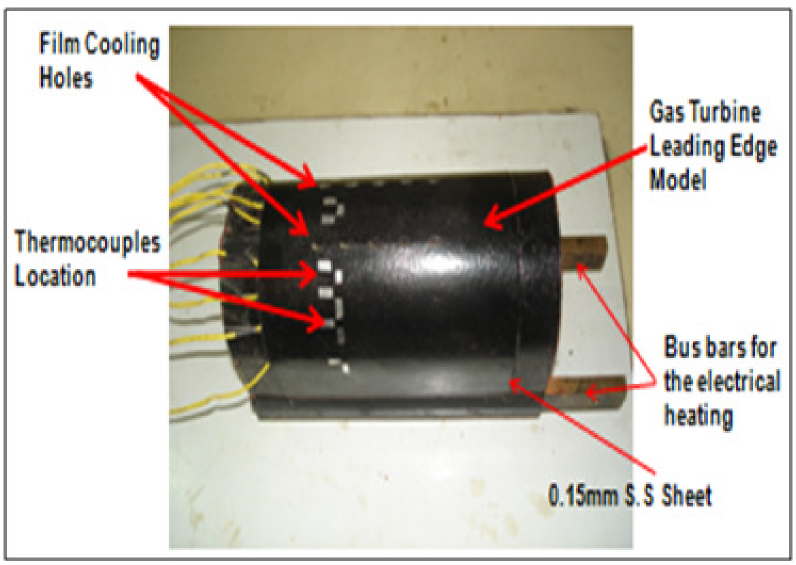

Fig. 2: Compound Angled Leading Edge Model

The Fig.2 shows gas turbine leading edge models with the thermocouples, etc. respectively and the varied coolant hole angles are shown in Fig.3 and the models are prepared to withstand the experimental conditions.

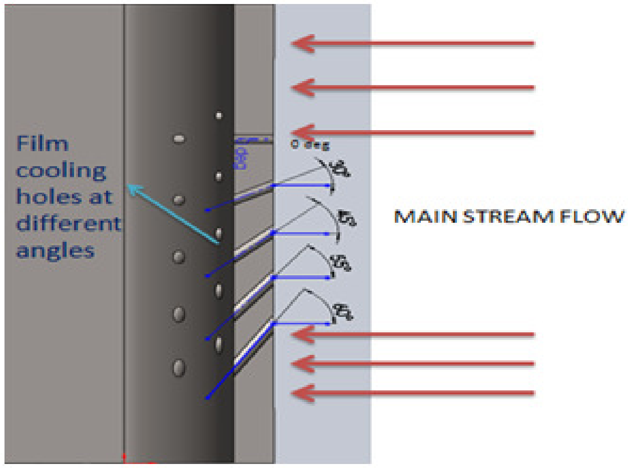

Fig. 3: Details of 0, 30, 45, 55, and 60 Deg. Hole Inclination Angles w.r.to Stream Line Direction

2.2 Experimental Setup

Gas turbine leading edge model is mounted in the test module, consisting of rectangular duct with a size of 320mm x 230mm x 700 mm. Experimental test facility consists of compressed air unit, settling chamber, air filter, control valve, orifice meter and rectangular duct test sections where gas turbine leading edge models are placed. Air is selected as working fluid for both mainstream and coolant. Main stream flows through the settling chamber to the test section. The main flow is controlled by the gate valve placed much ahead of settling chamber. The coolant air to the model is passed through the heat exchanger, where the controlled liquid nitrogen is used cool the coolant air to have the required coolant temperature. The static and total pressures of mainstream flow to the inlet of test section are measured and maintained to have the required Reynolds number. The coolant flow passing through the orifice meter is also maintained by monitoring the upstream and differential pressures across the orifice meter. The required coolant flow is maintained to have the blowing ratios of 1.00, 1.50, 2.00 and 2.50. The mainstream and coolant temperatures are monitored and maintained to have the required density ratio. To measure the pressure and temperature of main stream and coolant air, pressure ports and thermocouples are incorporated at inlet and outlet of rectangular duct and inlet coolant chamber. Flir make Infra-red Camera is used for the non-contact type temperature measurement of the test surface as shown in Fig.4. The calibrated reference thermocouples are placed on the test model to correct thermo gram test surface data obtained by the Infra-red camera. Pressure net scanner is used for measuring pressures from pressure ports and the Fluke data acquisition is used to measure the temperatures of Thermocouples For these experiments.

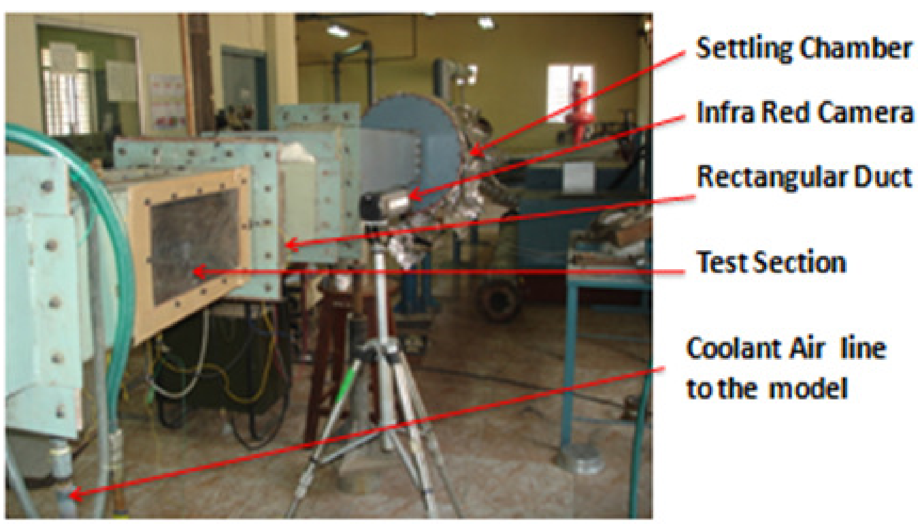

Fig. 4: Experimental Test Setup

The air is drawn from the centralized air compressor facility of NAL, with the continuous pressure of 60psi. The gate valve is used to maintain experimental pressure at the inlet to the test rig. The rectangular test section is fully tightened to withstand with high pressure. The test section is made with large opening to view the model by infrared camera through transparent sheet and also the provision is made for coolant air supply to the cooling chamber.

2.3 Film Cooling Effectiveness Measurement.

The main stream air is allowed to flow over the test surface and the coolant flow is passed through the coolant chamber, which ejects through the film cooling holes. The required coolant flow is maintained to have the blowing ratios i.e. the coolant to main stream mass flux ratio of 1.00, 1.50, 2.00 and 2.50. The coolant temperature is maintained around 231K by controlled liquid nitrogen flow to the heat exchanger to have the required density ratio i.e. the coolant to main stream density ratio of 1.30. The main stream air at ambient temperature coming from the centralized compressor facility is maintained at 24mm of H_2O to have the Reynolds number of 100000 based on the leading edge diameter. The wall temperature is measured using non-contact type infra-red camera after the steady state main stream and coolant flows have been achieved. The film effectiveness measurements are made with the mainstream at ambient temperature, coolant temperature and with the wall temperature measured as adiabatic wall temperature, as the surface is unheated and well insulated. The local wall temperature is mixture temperature of the coolant and mainstream. Thus, the film effectiveness is found using the following relation.

$$\eta = \frac{(T_m - T_w)}{(T_m - T_c)} \quad (1)$$

The blowing ratio (BR) is maintained using the ratio of Coolant mass flux ratio to mainstream mass flux ratio the relation is given by

$$B.R = \frac{(\rho c * Vc)}{(\rho \infty * V \infty} \quad (2)$$

The main stream flux ($\rho_\infty V_\infty$) was estimated from the measured mainstream velocity and the estimated mainstream density based on the static pressure and total temperature measured in the test section. The coolant mass flux ($\rho_c V_{c)}$ was estimated by dividing the measured mass flow by the total cooling hole area based on the inlet diameter of the film cooling hole. After establishing the required blowing ratio, the coolant temperatures and the model surface temperatures measured by the thermocouples were continuously monitored.

An uncertainty analysis performed indicates that the uncertainty in the calculated blowing ratio was $\pm$ 0.06 based on mainstream velocity uncertainty of $\pm$0.05 m/s and coolant velocity uncertainty of $\pm$0.6m/s.

3. COMPUTATIONAL DETAILS OF PRESENT STUDY

Film cooling effectiveness data is calculated experimentally for different B.R of 1.0, 1.50, 2.0 and 2.50 for the compound angled blade leading edge configurations with 0, 30, 45, 55 and 60 deg. hole angle of coolant holes with respect to stream line direction. The same has been tried numerically using ICEM CFD meshing and Ansys fluent solver software and to have the comparison with the experimental data.

3.1 Geometry of the Computational Model

The model consists of leading edge of gas turbine blade with the computational domain. Computational domain with the size of 320 x 210 x 300 mm with leading edge outer diameter of 89mm and inner diameter of 65mm, having a row of five film cooling holes of 4mm dia. with a hole pitch of 21mm is generated. Computational model is prepared as per experimental test section and only half of length of the test section is taken and symmetry portion of the leading edge is taken to avoid more number of element cells and analysis running time. Computational model is prepared using solid works and imported in ICEM CFD for meshing purpose using blocking technique and the computational domain is shown if the Fig. 5.

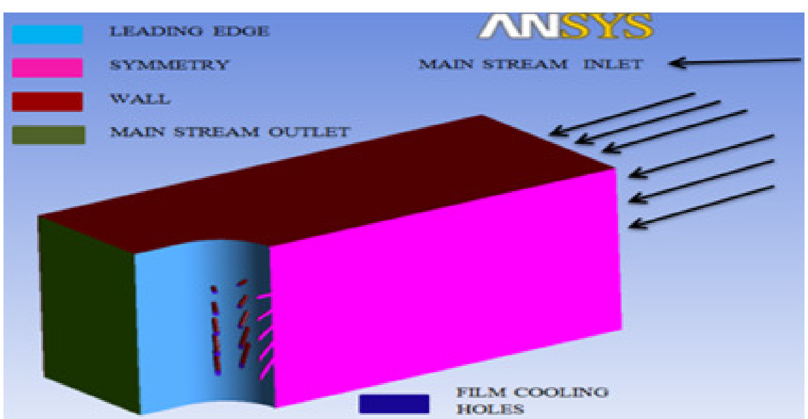

Fig. 5: The Computational Domain

The mesh is constructed for the considered geometry of the domain. The preprocessor, solver and Postprocessor modules are employed by ANSYS 14 Fluent. The output mesh file obtained is in .msh format which is used to run in fluent. Hexa type mesh is used for main stream till leading edge of the fluid region in order to obtain highest accuracy. And O grid mesh is used for the coolant flow. Quality is obtained above 0.55 as shown in Fig 6(a). The assumed y^+ value of 30 near wall adjacent for the turbulence model the obtained $\delta y = 0.158$. In total 449682 hexahedral cells have obtained for the entire geometry. The computational model after the meshing with grid cells is shown in the Fig.6(a)**.** The film coolant hole is meshed separately using O grid technique. At the inlet and outlet coolant hole is obtained with O grid. Coolant holes are attached to the blade leading edge model with mesh connectivity by using splitting technique near the hole fine mesh. The coolant hole with the coolant flow meshed with O grid is shown in the Fig.6(b).

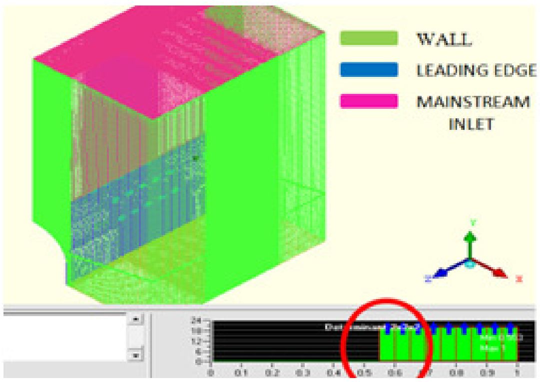

Fig. 6(a): Grid Generation of Computational Model

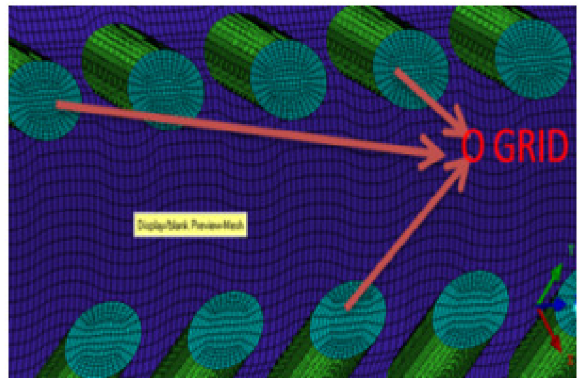

Fig. 6 (b): O Grid Generation of Cooling Holes

3.2 Boundary Conditions

TABLE 2: Boundary Conditions

Sl.No.	PART	BOUNDARY TYPE
1	LEADING EDGE	WALL
2	COOLANT INLET AND MAIN STREAM INTLET	PRESSURE INLET
3	COOLANT OUTLET	INTERIOR
4	MAIN STREAM OUTLET	PRESSURE OUTLET
5	SYMMETRY	SYMMETRY
6	WALL	WALL

In the process of finding the solution for film cooling effectiveness the k-omega SST, k-epsilon standard and k-epsilon realizable turbulence models are tried to get the solution for this model, among which the k-epsilon realizable turbulence model gives the better solution, which is nearer to the experimental values. Hence, the k-epsilon realizable turbulence model is used for all the blowing ratios for the CFD simulation. The type of boundary conditions applied in the computational model is shown in the Table 2. The boundary condition values are used same as per experimental test conditions.

4. RESULTS AND DISCUSSION

The Film temperature obtained on the leading edge surface by both experimentally evaluated and numerically simulated by varying the blowing ratios from 1.0 to 2.50. By using this film temperature values on the leading edge surface, the film cooling effectiveness is calculated theoretically along streamwise direction for all the blowing ratios. The temperature and velocity contours obtained by CFD are shown in the Fig.7 to Fig.10 for varying B.R.

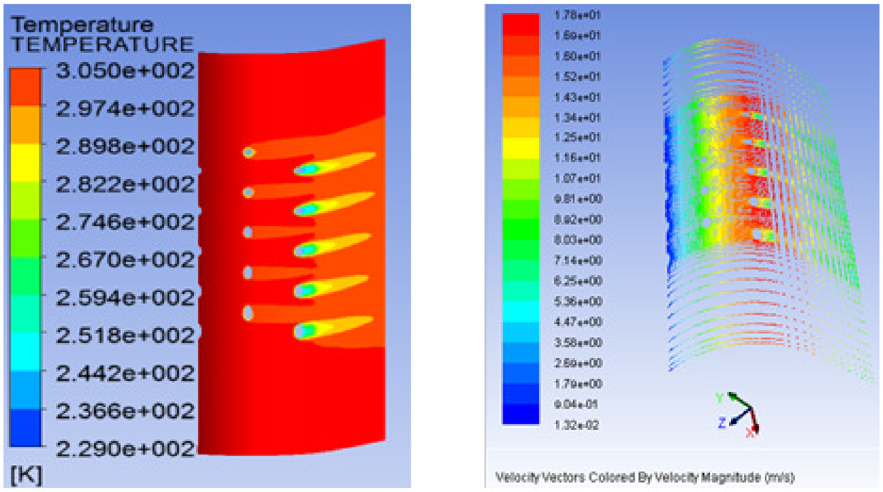

Fig. 7: Temperature and Velocity Contours, B.R=1.0

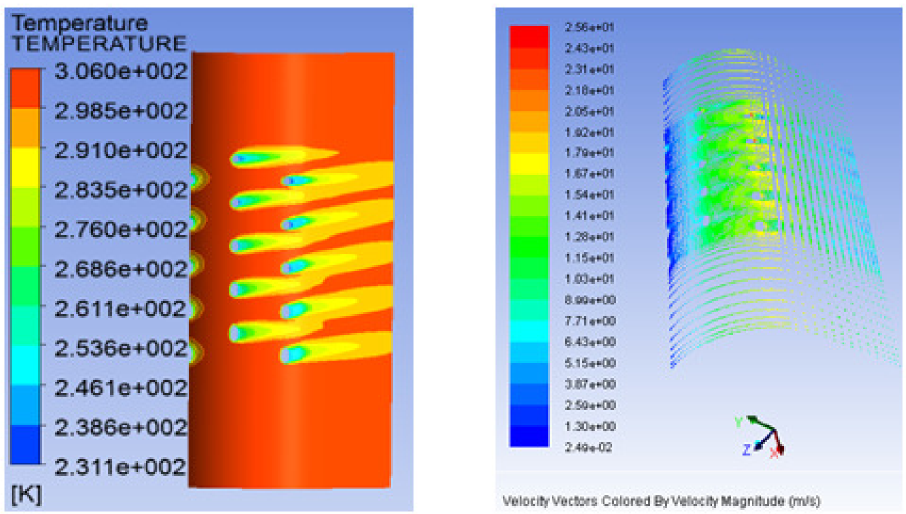

Fig. 8: Temperature and Velocity Contours, B.R=1.50

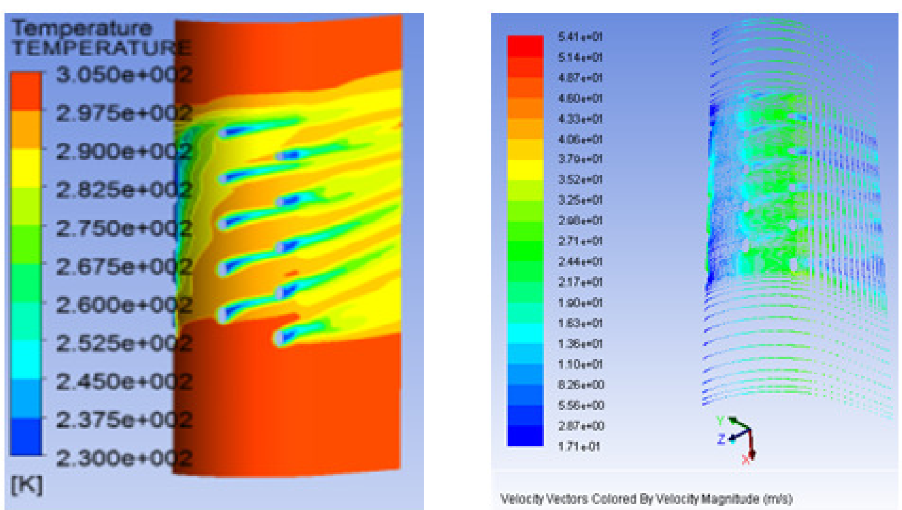

Fig. 9: Temperature and Velocity Contours, BR=2.0

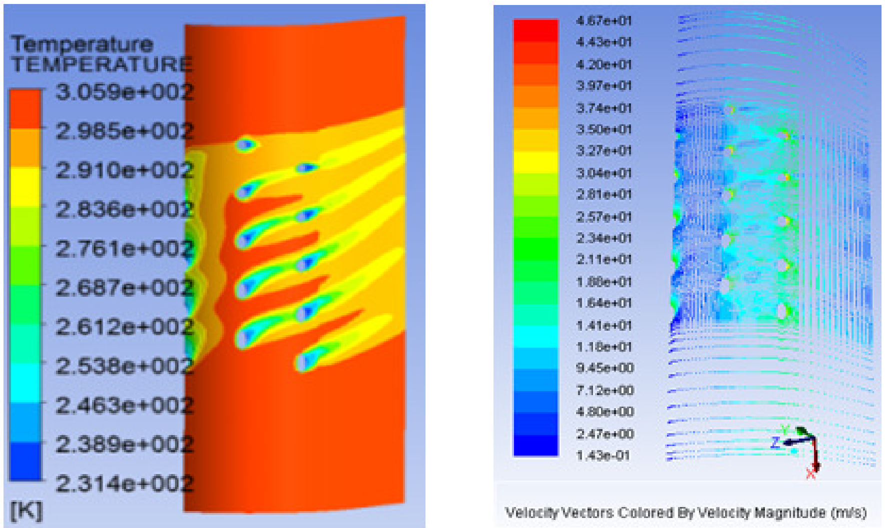

Fig. 10: Temperature and Velocity Contours BR=2.50

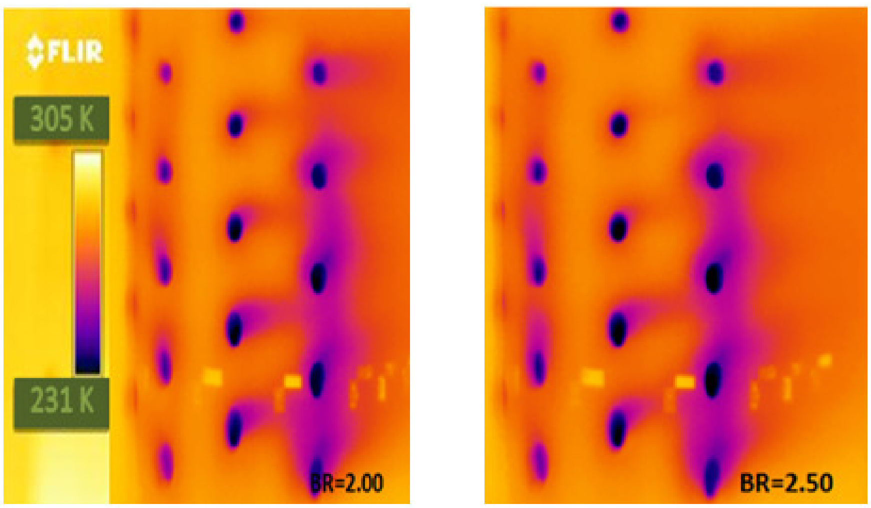

Fig. 11: Infra-Red Thermal Image for the Film Cooling Effectiveness at BR=2.0 and 2.50

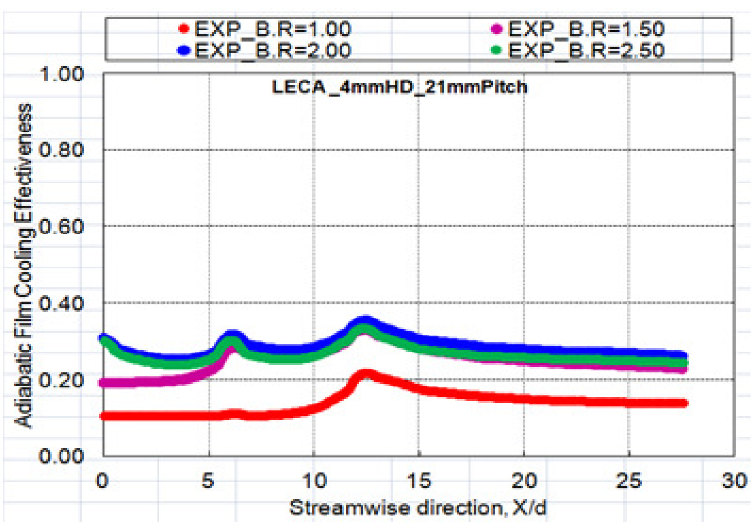

Fig. 12: Experimentally Evaluated Averaged Adiabatic Film Cooling Effectiveness, BR =1.0 to 2.50

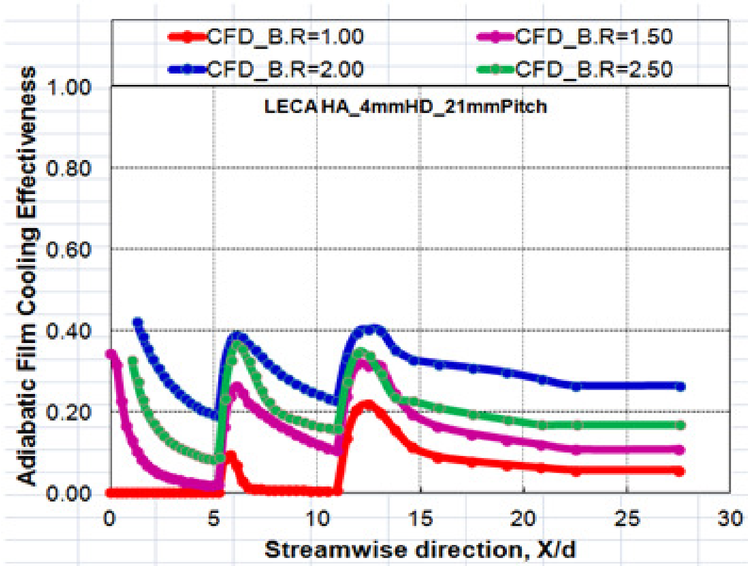

Fig. 13: Numerically Evaluated Averaged Adiabatic Film Cooling Effectiveness, BR=1.0 to 2.50

The Fig.11 shows the infrared thermal image of leading edge surface at B.R of 2.0 and 2.50. From the Fig.9 and Fig. 10 the flow patterns from CFD and temperature contour is same as experimental thermal image Fig.11. The typical experimental film cooling effectiveness and numerically investigated results at the B.R from 1.0 to 2.50 are shown in the Fig.12 and Fig.13. Both the experimental and numerical data shows the increase in cooling effectiveness with the increase in blowing ratio from 1.0 to 2.0, and from 2.0 to 2.50 the effectiveness decreases. The higher cooling effectiveness is observed at BR=2.0 due to the mass flow increase in coolant holes at higher B.R. Hence the optimized blowing ratio for this configuration can be considered as 2.0.

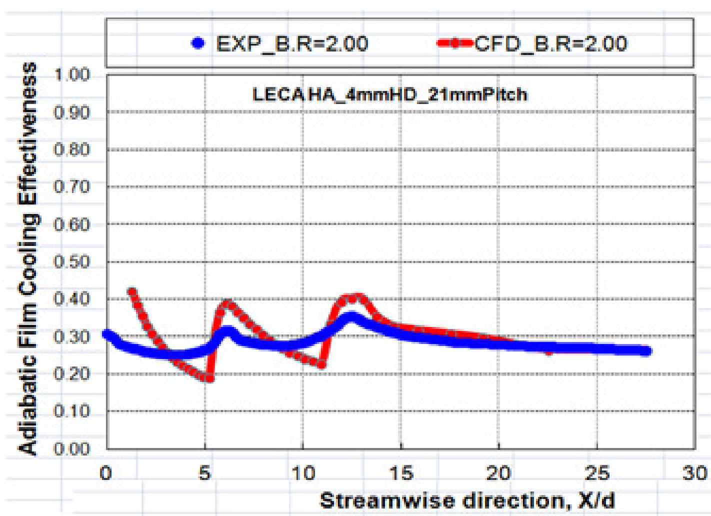

Fig. 14: Comparative Plot of Experimentally and Numerically Evaluated Averaged Adiabatic Film Cooling Effectiveness, BR=2.0

From Fig.13, the numerically evaluated film effectiveness shows the increase with the increasing B.R as that of experimental values. The Fig.14, shows the comparative film cooling effectiveness at BR=2.0 by both experimentally and numerically. The shift in the curve is observed due to the conduction process taking from the test chamber to the leading edge surface.

5. CONCLUSIONS

The present study deals with experimental and numerical investigation of adiabatic film cooling for gas turbine blade leading edge compound angled model having the different cooling hole injection angles of 0, 30, 45, 55 and

60degrees with main stream direction. From the experimental evaluation, the cooling effectiveness shown the increase with the increasing B.R and it is observed higher value of film cooling effectiveness at the B.R=2.0 among the considered blowing ratios and further increase in the blowing ratio above 2.0, there is no increase in film cooling effectiveness for this model.

From numerical investigation, film effectiveness also shown increase with increasing blowing ratio and the values found were nearer to the experimental values with the same flow patterns. Among the three considered turbulent solver models k-epsilon realizable turbulence model showed flow conditions as per the experiments. From both the experiments and numerical analysis, it is found that for the lower blowing ratios the film cooling effectiveness is very low and as the increase in blowing ratio the effectiveness increases upto B.R=2.0 and decreases above B.R=2.0 and hence the optimized blowing ratio can be considered as 2.0 for this type of configuration.

6. ACKNOWLEDGMENTS

The authors wish to thank Manjunath P., Head, Propulsion Division and Director, NAL, Bangalore for permitting this work to be carried out at NAL. The authors would also like to thank Mr. Felix J, Mr. Maria Arockiam, Mr. Sanmuganantham and Mr. Ravishankar N for their help in fabrication, assembling and assistance to conduct experiments and for their valuable suggestions during this course of work.

7. NOMENCLATURE AND SYMBOLS

B.R	-	Blowing Ratio
D.R	-	Density Ratio
ρc	-	Coolant Density (kg/m^3)
ρm	-	Mainstream Density (kg/m^3)
Vc	-	Coolant Velocity (m/s)
V_m	-	Mainstream Velocity (m/s)
η	-	Film cooling effectiveness
T_m	-	Mainstream Temperature (K)
T_c	-	Coolant Temperature before Injection (K)
T_w	-	Wall Temperature (K)
W	-	Width of stainless steel sheet (mm)
h	-	Heat transfer Coefficient (w/m^{2}k)

8. REFERENCES

[1] Sang woo Lee and Yong Beom kim, Film Cooling Jets With Compound Angled Orientations, ASME journal of turbomachinary,Vol. 119/310, 1997, pp. 310 – 3199.

[2] Srinath Ekkad and Chin Han JE, Recent Developments in turbine Film Cooling, International journal of rotating machinery, Vol. 7, No. 1, 2001, pp. 21 – 40.

[3] Bogard D. G and Thole K. A, Gas Turbine Film Cooling, Journal of propulsion and power,Vol. 2, No. 2, 2006, pp. 250 – 269.

[4] Govern M. C and Leylek J. H, Detailed analysis of film cooling physics, ASME journal of Turbomachinary, vol 122/113, Jan 2000.

[5] Je-Chin Han., Recent Studies in Turbine Blade Cooling, International Journal of Rotating Machinery, 10(6): 443-457.