Middle-East Journal of Scientific Research 25 (1): 90-94, 2017
ISSN 1990-9233

DOI: 10.5829/idosi.mejsr.2017.90.94

Computation Studies on End-Wall Film Cooling from a Single Row of Film Holes for the Three Different Geometry at Different Orientations

D. Ravi and K.M. Parammasivam

Department of Aerospace Engineering, MIT, Anna University, Chennai-44, India

Abstract: Film cooling is an efficient technique of bringing down thermal load stresses in gas turbines which undergoes very high pressure and temperature cycles. Film cooling involves bleeding coolant air through discrete holes on the blade end-wall which form a thin film layer between the mainstream flow and the blade surface. The thin film of cold air product the hot combustion gas from touching the turbine blade directly there by it gives increasing blade life. Gas turbine blade surfaces and end-walls are convectively cooled by film cooling, whose film cooling effectiveness forms the measure for assessing the efficiency. The influence of a single row of film holes at different orientations were computationally investigated with three specific shapes taken into consideration with three cases of inclination for each shape. The three shapes taken into consideration are Circular, Elliptical horizontally and Elliptical vertically. Three cases of inclination of tilt of coolant stream were chosen namely Normal to flow, adverse tilt and favorable tilt. The software used is GAMBIT (to generate the mesh with boundary layer conditions) and FLUENT ANSYS (for the flow contours and subsequent result generation). The results obtained from the CFD analysis is used to conclude the most effective shape and tilt from the observed cases, the values are plotted on a graph for comparison. The elliptical cylinder with adverse tilt was found to be the best film cooling orientation and hence this geometry is recommended from the cases taken into consideration.

Key words: Gas turbine · Film cooling · High pressure · Blade tips · Air product · Effective shape

INTRODUCTION

Generally the gas turbine blade surfaces and its end-walls are convectively cooled by film cooling, whose effectiveness forms the measure for assessing the efficiency. Film cooling is applied to nearly all of the external and internal surfaces associated with the airfoils that are exposed to the hot combustion gasses such as the leading edges, pressure side, suction side, main bodies, blade tips and end-wall junction. The development and structure of the secondary flow downstream of a single row of holes with compound angle orientations producing film cooling at high blowing ratios were discussed in the literature, such as in Ligrani and Lee [1]. This film cooling configuration is important because similar arrangements are frequently employed on the first stage of rotating blades of operating gas turbine engines.

Yuen and Martinez [2] studied the film cooling effectiveness experimentally on a cylindrical hole with three different streamwise angle of 30, 60 and 90 in a flat plate test setup with a zero pressure gradient. The hole length to diameter ratio (L/D =4) maintained constant for all the three geometries. The blowing ratio maintained between 0.33 to 2 and the Reynolds number with respect to free stream velocity and film whole diameter was 8563. The local and lateral averaged effectiveness are calculated across the central hole. The existing results are compared with the experimental results got by other researchers and the influence of variations in injection angle is presented.

Mayhew and Baughn [3] investigated the performance of film cooling effectiveness on a flat plate in the presence of low and high freestrem turbulence is studied using the liquid crystal thermography. It contributes high-resolution color images which clearly show how the free stream turbulence spreads the cooling air around the surface where we want to protect. The different blowing ratios are followed for a model with three straight holes with three time of diameter pitch, maintaining with density ratio of nearly one.

Ozturk [4] investigated the the analysis of heat sinks of Central processing unit by computation fluid dynamics methods.

Corresponding Auhor: D. Ravi, Department of Aerospace Engineering, MIT, Anna University, Chennai-44, India.

Bogard and Thole [5] Investigated the life time of the gas turbine engines is strongly dependent with respect to component temperatures. Film cooling is employed to reduce the gas turbine engine component temperature of combustor, turbine airfoils and endwalls. Film cooling is provided to nearly all of the external surfaces associated with the gas turbine engine components.

Giovanna Barigozzi and Giuseppe Franchini [6] Investigated the end-wall film cooling through four rows of cylindrical holes with conical expanded exits. Different area ratio has been compared with two end-wall geometries. The experimental test have carried out at M=0.2. By thermo chromic liquid crystals technique they might found adiabatic effectiveness distribution.

Alok Dhungel and Srinath [7] Analysed film cooling performance for a row of cylindrical holes each supplemented with two symmetrical anti vortex holes, it branch out from the main holes. The Reynolds number is 9683 based on the free stream velocity and film whole diameter. The four different blowing ratios were 0.5, 1, 1.5 and 2 also using the transient IR thermography technique. Film cooling effectiveness and heat transfer coefficient are obtained from a single test.

Sundaram and Thole [8] Investigated the gas turbine engine airofoil leading edge region along with the endwall of a steator vane influence high heat transfer rates from the formation of horseshoe vortices. They studied the trench configuration film cooling it provides overall better cooling relative to the no trench.

Colban and Thole [9] studied the shaped film cooling holes based on their investigation they developed emprical correlation for calculating laterally averaged film cooling effectiveness on a flat-plate downstream of a row of film cooling holes.

Kamil Abdullah and Funazaki [10] presented the numerical and experimental investigation of multiple film cooling hole which focuses on shallow hole angle at 20°. In this film cooling configuration an in-line hole of 20 cooling hole have been considered in the present study. The Reynolds number is 6,200 and blowing ratio is 1.0. 3D-LDV device was used to capture the experimental velocity flow field. The computation investigation was carried out through RANS analyses.

- The meticulous literature survey reported above exposed that various researchers have studied the flow and thermal load characteristics of a single hole with different geometrical shape, single row of holes and multiple rows of holes for the end wall film cooling of gas turbine blades. Simple and compound angled holes were studied at various blowing ratios. However, the influence of single row of film coolant holes at different alignment was not studied in depth. Hence, the computational investigations were carried out on a basic domain where the end-wall surface was cooled by a one row of basic film cooling holes of diameter'd', whose axes made an stream-wise angle of 30° with the end-wall surface. Based on the air properties the Reynolds number was estimated at slightly greater than 2e+5, it shows that the flow was turbulent at the inlet. The computations were performed using the tool Ansys Fluent 15.0, for a fixed blowing ratio (M) of 0.6. Atmosphere air was considered as the mainstream hot fluid and the film coolant injected through the holes.

Computational Domain: The main scope of the project is to create a model in gambit and then analyze it using ANSYS FLUENT. For any simulation, modeling software has to be used to initially create the model, in our case Gambit was used and the dimensions were specified according to the values that were interpreted from the theoretical papers used.

There are three different flows we take into consideration we initially consider to computationally investigate the influence of a single row of film holes at different orientations.

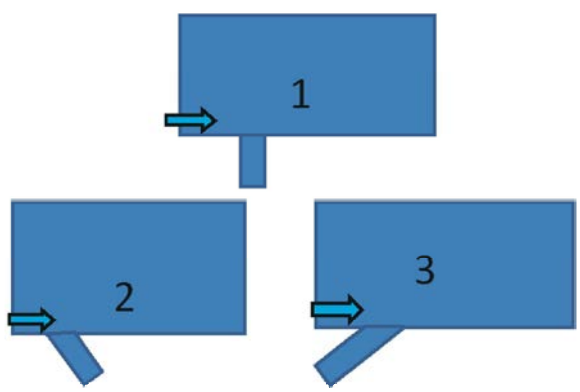
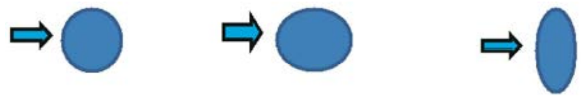


Fig. 1: Schematic diagram of coolant flow diagram
1. Normal to flow 2. Adverse tilt 3. Favourable tilt

Three different whole shapes are being considered for the present study. End wall film cooling effectiveness will be compared and reported. There are 3 different shapes that are taken into considerations for which the study is done individually.

1. Circular 2. Horizontally Elliptical 3. Vertically Elliptical



Therefore the total number of cases under going study is

1. Circular cylinder	2. Elliptical Cylinder (E1) – Thin (normal to flow)
• Normal to flow (N)	Normal to flow (N)
• Adverse tilt (A)	Adverse tilt (A)
• Favorable tilt (F)	Favorable tilt (F)

3. Elliptical Cylinder (E2) – Thick (normal to flow)
- Normal to flow (N)
- Adverse tilt (A)
- Favorable tilt (F)

Circular cylinder (C) as the main inlet dimensions for the jet hole and have the flow of the cold air be normal to the main flow which consist of hot air. This is the first case taken in to consideration for the study of the film cooling effectiveness. The second case is with the same cylindrical dimensions at the base of the jet hole and then the jet hole is tilted in a direction adverse to the main flow. The third case is where the base dimension is cylindrical and then the jet hole is tilted in a direction favorable to the flow of the main stream.

The fourth case is when the vertically elliptical cylinder is considered as the inlet hole to guide the flow of the jet path. Again, in the same manner as the previous one, three cases are considered which are, Normal to flow – which has a jet of the air coming from the compressor flowing normally to the other passing flow of the mainstream hot gases, then we have the fifth case, the Adverse tilt – where the jet is kept at a tilt angle to push the jet against the flow the passing mainstream flow and the sixth case, the Favorable tilt is used where the jet is placed in an angle so that the flow of the jet is towards the same line as the passing mainstream fluid so that it is not hindered by the cross current.

Table 1: Geometrical parameters of the gas turbine cascade

Parameter of the blade	Units	Value
Reynolds number, *Re*	-	24.71e+05
Blowing ratio, *M*	-	0.6
Temperature ratio, T_m/T_c	-	1.034
No. of film hole rows	-	1
Film hole diameter, *d*	mm	4
Film hole pitch, *P*	-	2*d*
Film hole length, L/*d*		10

The seventh case consists of a horizontally elliptical jet inlet with a normal flow going against the flow of the mainstream inlet, the eight case is that of a horizontally elliptical jet inlet with adverse flow where the flow is against that of the mainstream flow and finally the ninth case where the horizontally elliptical jet inlet flows along that of the mainstream flow.

End wall film cooling effectiveness is to be compared and reported by using GAMBIT (to generate the mesh with boundary layer conditions) and FLUENT ANSYS (for the flow contours and subsequent result generation).

Computational Mesh Details: The construction of the domain for mesh was done by using the dimensions taken from the previous literature surveys.

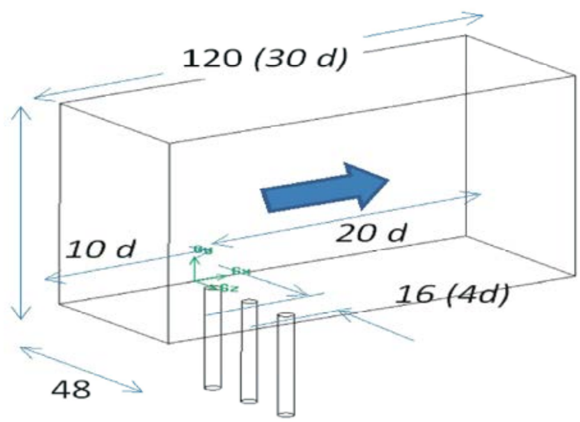


Fig. 1: Computational Domain- Jet Normal to Flow

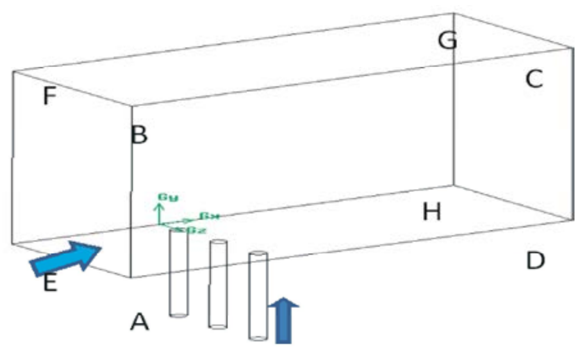


Fig. 2: Computational Methodology

Boundary type	Surface chosen
Symmetry	Faces ABCD, EFGH
Wall	FGCB
Interface	AEHD
Mainstream inlet	AEFB
Mainstream outlet	HDCG
Jet inlet	3 circular faces at the end of the jets

For the boundary layer mesh details the growth factor is kept higher than 1 so that the mesh count increases as it goes towards the plenum base and the jet wall sides. The main reason for this is so that the mesh can analyse more of the change pattern in the flow from the jet hole as the minstream causes it to form a film layer.

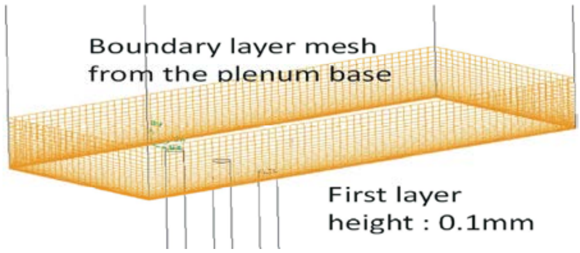


Fig. 3: Boundary Layer Mesh Details from Plenum Base

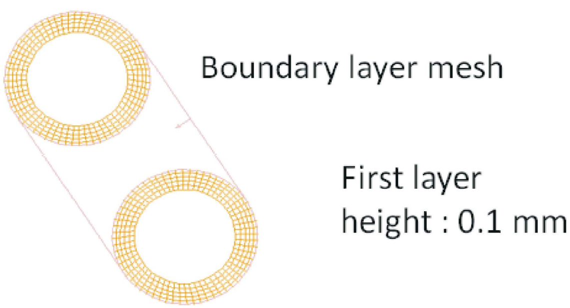


Fig. 4: Boundary Layer Mesh Details from Jet Walls

The element type of mesh used hexahedral Non-conformal mesh.The no of elements in the jet is higher than the mainstream.

Interface type boundary condition was used between jet exit and plenum bottom.

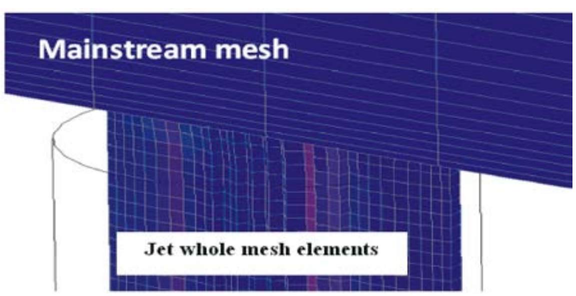


Fig. 5: Hexahederal Type Non-conformal Mesh Was Used

RESULTS AND DISCUSSIONS

The outcomes from the computational fluid dynamic (CFD) analysis of the end-wall surface of the film cooling applicable to gas turbine blade cascade, with film cooling hole positioned at different configuration, are presented here in this section.

Data Reduction: The flow Reynolds number is calculated based on the air density, coolant velocity and its diameter at the test section inlet it is defined as below:

$$Re = \frac{\rho_m Cv\ d}{\mu_m} \tag{1}$$

The adiabatic film cooling effectiveness is estimated using formula given below.

$$\eta = \frac{T_{m,in} - T_w}{T_{m,in} - T_{c,in}} \tag{2}$$

where $T_{m,\,in}$ is the mainstream fluid inlet temperature; T_w is the surface wall temperature; $T_{c,in}$ is the coolant inlet temperature. The pitch wise averaged film cooling effectiveness is calculated as the arithmetic average of the local values and is denoted as $\bar{\eta}$.

The boundary conditions used for the CFD analysis were taken with the blowing ratio of 0.6 and inlet velocity of mainstream and coolant to be 6 m/s and 10 m/s respectively. The inlet temperature of the mainstream temperature as 310 K and the coolant was 300 K.

CFD Results

Contours of Temperature on Chosen Surfaces:

Circular Cylinder – Normal (C-N)

Blue regions indicate the coolant dominant regions whereas the Red zone indicates the hot fluid from the main stream gas; the other colors indicate the missing of the hot and cold fluid.

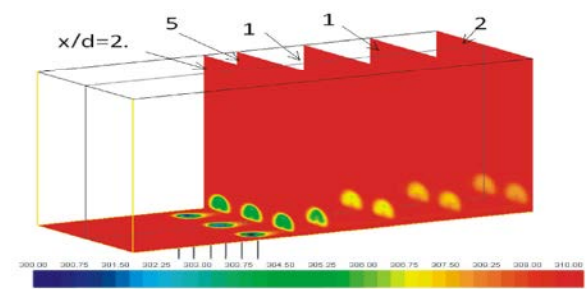


Fig. 6: Contours of temperature on chosen surfaces

Contours of Velocity Magnitude on Chosen Surfaces:

Circular Cylinder – Normal (C-N)

This plot shows the evolution of flow in the mainstream direction. The flow lifts off from the bottom surface of the channel. The movement of the core of the vortex can be observed by the dashed line shown.

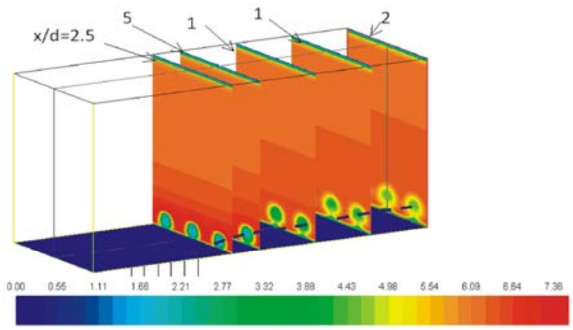


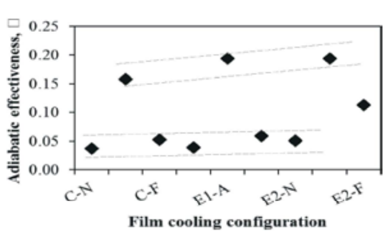
Fig. 7: Contours of Velocity Magnitude on Chosen Surfaces

Highest values of η are observed close to the hole row location, where the hot mainstream fluid and injecting coolant interact. The CFD results are then taken and plotted on a table

Table 2 The results from the computational fluid dynamic analysis of the end-wall film cooling of a gas turbine blade cascade, with film cooling row positioned at different stream Wise locations.

Geometry	Orientation	Notation	Adia effec
Circular	Normal	C-N	0.0377
	Adverse	C-A	0.1582
	Favourable	C-F	0.0525
Ellipse thin	Normal	E1-N	0.0395
	Adverse	E1-A	0.1938
	Favourable	E1-F	0.0592
Ellipse thick	Normal	E2-N	0.0509
	Adverse	E2-A	0.1941
	Favourable	E2-F	0.1129

Graph 1 Show the results from the computational fluid dynamic analysis of the end-wall film cooling configuration applicable to first stage of stator of gas turbine blade cascade, with film cooling row positioned at different stream wise locations.



CONCLUSION

From the present CFD analysis the following conclusions were arrived at,

- Film holes with adverse tilt were found to produce higher effectiveness.
- E2-An ie. the Elliptical cylinder with adverse tilt showed highest value. This geometry is identified as the best for film cooling configuration (among the presently tested ones) and hence it is recommended.

REFERENCES

1. Ligrani, P.M. and J.S. Lee, 1996. Film Cooling from a single Row of Compound Angle Holes at High Blowing Ratios, International Journal of Rotating Machinery, 2: 259-267.
2. Yuen, C.H.N. and R.F. Martinez-Botas, 2003. Film cooling characteristics of a single round hole at various streamwise angles in a crossflow: Part I effectiveness, International Journal of Heat and Mass Transfer, 46: 221-235.
3. Mayhew James, E. and James W. Baughn, 2003. The effect of free stream turbulence on film cooling adiabatic effectiveness, International Journal of Heat and Fluid Flow, 24: 669-679.
4. Ozturk, E., 2004. CFD analysis of heat sinks for CPU cooling with FLUENT, MSc Thesis, Graduate School of Natural and Applied Sciences, Middle East Technical University.
5. Bogard, D.G. and K.A. Thole, 2005. Gas Turbine Film Cooling, AIAA, Journal of Propulsion and Power.
6. Barigozzi, G and G. Franchini, 2007. End-Wall Film Cooling Through Fan-Shaped Holes With Different Area Ratios, ASME J. Turbomach., 129: 212-220.
7. Dhungel Alok and Srinath V. Ekkad, 2009. Film Cooling From a Row of Holes Supplemented With Antivortex Holes, ASME J. Turbomach., 131: 021007-9.
8. Sundaram, N. and K.A. Thole, 2009. Film-Cooling Flow fields With Trenched Holes on an Endwall, ASME J.Turbomach., 131: 041007-1-10.
9. Colban, W.F., K.A. Thole and D. Bogard, 2011.A Film-Cooling Correlation for Shaped Holes on a Flat-Plate Surface, ASME J. Turbomach, pp: 133.
10. Kamil Abdullah and Ken-ichi Funazaki, 2012. Experimental and Numerical Investigation on Flowfield of Film Cooling from Multiple Holes, Applied Mechanics and Materials Vols, pp: 2094-2099.