Effect of Coolant Jet Direction on Film Cooling Behavior

[1]Asst. Prof. Dr. Muwafaq Shyaa Alwan, [2]Asst. Prof. Dr. Amer M. Al-Dabagh, [3]Asst. Lect. Omar Hassan Hameed
[1,3]Al-Iraqia University, Collage of Engineering, Baghdad, Iraq
[2]University of Technology, Baghdad, Iraq

Abstract

Numerical investigations were done on a flat plate in order to predict the flow behavior, vortex type, and vortex generated at holes and holes downstream area, and the interaction between hot mainstream and coolant jet. Two type of singular coolant air jet were studied; the first jet hole inclined at 30º in the direction of hot air stream. The second holes are lying in the opposite direction at 30º. All simulation were conducted at blowing ratio of 0.5 and 1.5, the diameter of cooling hole is 4mm. The commercial CFD software FLUENT with standard $(k-\epsilon)$turbulent models was applied. The results showed that there are two large vortices have been detected, large counter rotating vortex pair generated from the hole rims and horseshoe vortices, both vortices have major effects on cooling performance. The results also showed that the reverse flow from backward injection hole creates pair of vortex similar to the kidney vortex created from forward injection hole, but in a plane parallel to the main stream.

Keywords: Film cooling, Jet direction, Kidney vortex, horseshow vortex.

1-Introduction

Turbine blades require better cooling technique to cope with the increase of the operating temperature with each new engine model. Film cooling is one of the most efficient cooling methods used to protect the gas turbine blades from the hot gases. Jet holes arrangement offers reliable technique help to improve the coolant effectiveness of the film cooling.

Film cooling primarily depends on the coolant-to-mainstream pressure ratio or can be related to the blowing ratio, temperature ratio (T_c/T_m), the film cooling hole location, configuration, and distribution on a turbine elements film cooling. In a typical gas turbine blade, the range of the blowing ratios is of about 0.5 to 2.0, while the (Tc/Tm) values vary between 0.5 and 0.85 Han and Ekkad[1].

Injecting behavior of two rows of film cooling holes with opposite lateral orientation angles have been investigated by Ahn et al. [2] in which four film cooling hole arrangements were considered including inline and staggered ones. Detailed adiabatic film cooling effectiveness distributions were measured using thermochromic Liquid Crystal to investigate how well the injecting covers the film cooled surface. They found that staggered opposite lateral arrangement shows best cooling performance.

Dhungel et al. [3] obtained simultaneously detailed heat transfer coefficient and film effectiveness measurements using a single test transient IR thermography technique for a row of cylindrical film cooling holes, shaped holes. A number of anti-vortex film cooling designs that incorporate side holes. They found that the presence of anti-vortex holes mitigates the effect of the pair of anti-vortices. Experimental and numerical investigations were done by Lu et al. [4 and 5] to measure and to predict the film cooling performance for a row of cylindrical holes. They used adiabatic film effectiveness and heat transfer coefficients were determined on a flat plate by using a single test transient thermograph technique at four blowing ratios of 0.5, 1.0, 1.5 and 2.0. Four test designs crescent and converging slot, trench and cratered hole exits, were tested. Results showed that both the crescent and slot exits reduce the jet momentum at exit and also provide significantly higher film effectiveness with some increases in heat transfer coefficients.

Dia and Lin [6] investigated numerically three film cooling configurations, (cylindrical hole, shaped hole, crescent hole). All holes were inclined at 35° on a flat plate. All simulations are conducted at blowing ratio of 0.6

and 1.25, length to diameter ratio of 4 and pitch-to-diameter ratio of 3. They use (RANS) equations, the energy equation, and two-layer $(k-\epsilon)$ turbulence models. For the numerical investigation the commercial CFD software FLUENT with standard $(k-\epsilon)$ turbulent models is applied. They found that the crescent hole exhibits the highest film cooling effectiveness among the three configurations both in spanwise and streamwise especially downstream of the interaction of the two holes.

Lee and Kim [7] evaluated the effect of geometric variables of a laidback fan-shaped hole on the film cooling effectiveness using a Reynolds-averaged Navier-stokes analysis. The shape of the laidback fan-shaped hole is defined by four geometric design variables: the injection angle of the hole, the lateral expansion angle of the diffuser, the forward expansion angle of the hole, and the ratio of the length to diameter of the hole. They concluded that the increase of the forward expansion angle makes a reduction of film cooling effectiveness, and the lateral expansion angle has the biggest impact among the four geometric variables on the spatially averaged film cooling effectiveness.

Numerical prediction of Alwan. [8] shows that the flow field structure of injected holes present vortices such as counter pair kidney vortex and horseshoe vortex have major effects on cooling performance, in which the strength of the kidney vortex decreases and the horseshow vortex was lifting up, leading to an improvement in the coolant performance. Therefore numerical model was suitable to design holes arrangement futures of film cooling system by introducing oriented holes row over single jet holes row.

Most literature focuses on the study of the effective parameters of film cooling for one row film holes in the forward direction with mainstream. There is no information available for the row of holes in the backward direction with mainstream flow, also lack information available for two rows of film cooling on forward direction. However, at the present work experimental and numerical investigations were done to evaluate the cooling performance (film cooling effectiveness and heat transfer coefficient) by using a single test transient IR thermograph technique for a different holes direction and at different blowing ratio.

2- Single jet film cooling high light

In film cooling used in gas turbines, the coolant is introduced to the hot gas stream of the turbine section as cross flow jets. The coolant forms a film layer over the turbine walls, to protect the surface from direct exposure to the hot gas stream. Different injection angles have been proposed with an optimum injection angle and are found to be (30º-35º) [71]. The behavior of the flow structure is generated as a result of two flows interaction affected the performance of film cooling; therefore details of flow structure associated with this interaction should be highlighted and discussed in details. To localize the effects and reduce the number of variables, the geometries of the blade is a simplified to film cooling over flat plate created by means of cooled air jet injected into the cross flow boundary layer through an isolated one jet hole inclined at 30º in the direction of hot stream. The second holes are lying in the opposite direction at 30º which represent the benchmark of jet with cross flow in the present discussion as shown in Figure (5.1). The interaction of the jets with cross flow in three dimensional domains is a complicated flow regime. To simplify the case and to make the flow recognizable and readable, the flow will be presented in two dimensions in a plane perpendicular and parallel to the cross flow at different plane location. These planes are captured from the computation approach using computer code fluent (12.1).

3- Numerical procedure

In the present study, air is taken as the working fluid and the flow characteristics are assumed to be steady flow, Newtonian fluid, incompressible fluid (Mach number=0.11), turbulent flow, three dimensional. FLUENT version (12.1), GAMBIT software and Auto CAD 2011will be used to create, grid for the system geometry. To localize the effects and reduce the number of variables, the geometries of the blade is a simplified to film cooling over flat plate created by means of cooled air jet injected into the cross flow boundary layer through an isolated one jet hole inclined at 30º in the direction of hot stream. The second holes are lying in the opposite direction at 30º which represent the benchmark of jet with cross flow in the present work, and with two blowing ratio (BR=0.5 and 1.5). The solution of the Reynolds Averaged Navier-Stokes and energy equations is obtained by using the FLUENT software. Fluent is based on an unstructured solver using a finite volume approach for the solution of the RANS equations. The system geometry consists of the box with dimensions (128x12x50) mm for the hot mainstream, box with dimensions (35x12x20) mm for coolant jet and tow different models of hole as shown in Figure (1). The system geometry is drawn by using (Auto CAD 2011 code). The diameter of cooling hole is 4mm.The coolant conditions were maintained the same in all cases and the mainstream flow rate was altered to change the blowing ratios. The Mainstream temperature was set at 322 K and the coolant temperature was set at 302 K. At the exit plane, pressure level was specified along with zero streamwise gradients for all other dependent variables.

The current study used the standard $(k-\varepsilon)$model for the simulating the turbulent flows in film cooling. The standard $(k-\varepsilon)$model is economical with reasonable accuracy for a wide range of turbulent flows and it is widely used in heat transfer simulation Versteeg and Malalasekera [14].There are some general guidelines to create a good mesh. These guidelines are shortly called rules of QRST standing for (Quality, Resolution, Smoothness, and Total cell count) Ozturk [15].The importance of quality parameter is the face alignment; it is the parameter that calculates skewness of cells. Elements with high skewness should be avoided. The way of checking whether the solution is grid independent or not is to create a grid with more cells to compare the solutions of the two models. Grid refinement tests for average static temperature on hot surface indicated that a grad size of approximately (2.5 million cell) provide sufficient accuracy and resolution to be adopted as the standard for film cooling system. The nodes near the test plate surface were adjusted so that average y+ value was about 20 near the test plate surface which is within the range of Jones and Clarke [16].The most significant factor to be monitored for the present model is the average static temperature on hot surface. When the average static temperature on hot surface value monitor converged it is unnecessary to go further on with the iterations and wait even if the residuals do not fall below the defined convergence criteria.

4- Results and Discussion

4-1 Forward injection

Figure (2) illustrates the main flow features of a single hole injected air in the direction of hot stream at two blowing ratio of 0.5 and 1.5. The jet cross flow interaction generates a turbulent flow, which is dominated by coherent motion. In general multiple vortex structures are produce where two large vortex structures have been detected, counter rotating vortex pair (CVP), and horseshoe vortices. CVP is the largest vortex appeared in the flow structure, which occurs at the interaction of coolant jet and hot main stream as shown in Figures (3&4). The vortex main structures are originated from the holes rims as shown in a front view direction which agrees with most literature [3, 72], while some literature claimed that these kidneys vortex may be generated within the hole core according to the magnitude of the momentum ratio. For a low momentum ratio of about (i. e. BR=0.5), the mainstream flow close to the test surface as shown in Figure (5). At low momentum, the jet streamlines seems to go towards the surface and the mainstream flow depart upward, therefore the mainstream push the jet towards the surface. For a high momentum ratio (i. e. BR=1.5), and since the momentum of the jet is relatively high, the jet is separated away into the mainstream. In both cases, the vortices are rolled up at the hole side edge and due to the shear between the jet and cross flow they continue warp and manifest downstream. As the vortex propagates downstream the hole, the vortex grows continually downstream as shown in Figure (3&4). Kidney -vortices have sense of rotation that acts as the coolant lifter. The jet lift-off phenomenon typically occurs at a momentum ratio (BR$>$ 0.5). For very low momentum ratio (i.e. BR=0.5), the circular hole produce slower jet vortices levels and the coolant stays attached to the surface, providing good film coolant effectiveness as shown in Figure (6).

In practice, BR of 0.5 is typically unachievable, as the available coolant pressure is unavoidably produces higher coolant flow rates. As the coolant leaves the surface, the rotation of the vortices also promotes entrainment of the hot gases down toward the surface. The combination of coolant lift-off and hot gas entrainment can seriously degrade the cooling effectiveness of the film cooling layer downstream as shown in Figure (2).

The flow upstream of the jet is strongly decelerated near the jet, which causes flow separation and the formation of the horseshoe vortex that warps itself around the jet. This is similar to the structure formed for a cylinder in cross flow. The vortex from hot stream moves around the jet at a constant radius from the center of the jet, but downstream it is involved to the jet by a decrease static pressure and then mixed with the jet. When comparing cooling effectiveness for BR= 0.5 and 1.5,as shown in Figures (7a&7b), CVP plays an important role in the contribution of jet lifting off, this can be seen clearly in the case of low momentum jet (BR=0.5) and the case of high momentum jet (BR=1.5), in which the horseshoe vortex is strongly influenced by high jet momentum, as a results of that the jet behaves like an adiabatic isolated tube surrounded by the horseshoe vortex, therefore the film cooling effectiveness decreased at hole downstream region.

From the above discussion, it can be noticed that CVP liftoff, entrainment hot air from the main stream by CVP and the horseshoe vortex have poor effect on the cooling performance especially at high BR. To avoid entrainments of hot stream, the CVP from the adjusting spanwise hole should be interacted with the adjusting one; this can be achieved only when traverse space to the diameter ratio (S/D) is less than 2.5. This approach violated the mechanical properties of the turbine blade, therefore by introducing row of staggered jet holes in the upstream region at (X/D=4), and (S/D=3), this may be enhance the cooling effectiveness and restore the mechanical properties of the turbine blade. This technique can be introduced as part of the present investigation.

4-2 Backward injection

To investigate the effect of backward injection isolated jet holes located in the opposite direction to the hot stream have been introduced. Figure (8) illustrates the main flow features of a single hole injected air in the direction opposite to hot stream at two blowing ratios of 0.5 and 1.5. The blockage created by the jet as it enters the main stream creates a local variation of pressure at the hole exit. The pressure of the injected air on the upstream side of the hole is elevated, thus locally reducing the jet velocity; and pushing up the hot stream depending on the blowing ratio. On the downstream side of the hole, the pressure falls and locally increases the exit velocity. As the cooled air penetrate into the hot stream, its momentum decreases up to the momentum of the main stream then bend back toward the surface causing lee vortex as shown in figure (8).

The pressure variation at the hole exit create a reverse flow where part of the cooled air exits in the direction tangent and normal to rims hole reducing the jetting effect at hole rims, which was responsible for creating kidney vortex as in the forward injection. This reverse flow creates pair of vortex similar to the kidney vortex downstream but in a plane parallel to the main stream as shown in figure (9), this vortex is sweeping near the surface and pushing away the horseshoe vortex where moderate and wider protection area are obtained at low and high BR around the hole area as shown in figure (10&11). In general, interaction and diffusion of the injected air with hot stream provides moderate air film temperature flowing near the surface and covering wide area. The preliminary results of CFD of backward injection are encouraging for implementing this technique for film cooling.

5- Conclusions

The present work has reached to the following conclusions:

1- Numerical prediction of the flow field structure for holes arrangement show that the vortices (counter pair kidney vortex and horseshoe vortex) both have major effects on cooling performance, in which the strength of the kidney vortex decreases and the horseshow vortex is lifting up, leading to an improvement in the coolant performance.

2- The reverse flow from backward injection hole creates pair of vortex similar to the kidney vortex created from forward injection hole, but in a plane parallel to the main stream.

REFERENCES

[1]Han, J.C. and Ekkad, S.V., " Recent Development in Turbine Blade Film Cooling", International Journal of Rotating Machinery, Malaysia, Vol. 7, No. 1, 2001, pp. 21-40.

[2] Ahn, J., Jung, I.S., and Lee, J.S., "Film cooling from two rows of holes with opposite behavior and adiabatic film cooling effectiveness", International Journal of Heat and Fluid Flow, Vol. 24, 2003, pp. 91-99.

[3] Dhungel, A., Phillips, A., Ekkad, S.V., and Heidmann, J.D., 2007, "Experimental Investigation of a Novel Anti-Vortex Film Cooling Hole Design", ASME IGTI Turbo Expo, Montreal, Paper GT 2007-27419.

[4] Lu, Y.,Dhungel, A.,Ekkad, S.V., and Bunker, R.S., 2007, "Effect of Trench Width and Depth on Film Cooling from Cylindrical Holes Embedded in Trenches", ASME Paper GT 2007-27388.

[5] Lu, Y., Dhungel, A., Ekkad, S.V., and Bunker, R.S., 2007, "Film Cooling Measurements for Cratered Cylindrical Inclined Holes", ASME Paper GT 2007-27386.

[6] Dia, P. and Lin, F., 2011,"Numerical study on film cooling effectiveness from shaped and crescent holes", Heat Mass Transfer, Vol. 47, PP. 147-154.

[7] Lee, K.D. and Kim, K.Y., 2011,"Surrogate based optimization of a laidback fan-shaped hole for film-cooling", International Journal of Heat and Mass Transfer, Republic of Korea, Vol. 32, PP. 226-238.

[8] Alwan, M. Sh., 2012," Experimental and Numerical Investigation of Film Cooling Thermal Performance for Staggered Rows of Circular Jet" PhD thesis, Mechanical Engineering Department , University of Technology.

[9] Holman, J.P. and Bhattacharyya, S., "Heat Transfer", Ninth Edition, New Delhi, McGraw-Hill, 2008.

[10] Ekkad, S.V., Ou, S., and Rivir, R.V., "A Transient Infrared Thermography Method for Simultaneous Film Cooling Effectiveness and Heat Transfer Coefficient Measurements from a single test", GT 2004-54236, Proceedings of ASME Turbo Expo 2004, Vienna, Austria.

[11] Ekkad, S.V., and Zapata, D., "Heat transfer coefficients Over a Flat Surface with Air and CO2 Injection Through Compound Angle Holes Using a Transient Liquid Crystal Image Method", ASME Journal of Turbomachinery Vol. 119, No. 3, 1997, pp. 580-586.

[12] Albert, J.E., Cunha, F. and Bogard, D.G., 2004, "Adiabatic and Overall Effectiveness for a Film Cooling Blade", ASME Paper GT2004-53998.

[13] Kline, S.J. and McClintock, F.A., 1953, "Describing uncertainties in single sample experimental", Mechanical Engineering, Vol. 75, pp. 3-8.

[14] Versteeg, H.K. and Malalasekera, W., 1996, "An introduction to computational fluid dynamics the finite volume method", Longman Group, London.

[15] 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.

[16] Jones, D.A. and Clarke, D.B., 2005, "Simulation of a wind-body junction experiment using the fluent code", DSTD-TR-1731, Australia.