Penerbit Akademia Baru

CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

Effects of Row Trench Holes Geometries on an Endwall Cooling

Open Access

Nor Azwadi Che Sidik[1], Ehsan Kianpour[2,*]

[1] Malaysia – Japan International Institute of Technology (MJIIT) , Universiti Teknologi Malaysia,Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia
[2] Department of Mechanical Engineering, Najafabad Branch, Islamic Azad University, Najafabad, Iran

ARTICLE INFO

Article history:
Received 11 February 2019
Received in revised form 1 June 2019
Accepted 26 June 2019
Available online 3 July 2019

ABSTRACT

Gas turbine industries try to extend the gas turbine engine performance. By using the well-known Bryton cycle, the combustor outlet temperature must be increased to have higher efficiencies. However, the turbine inlet temperature increment creates harsh environment for the downstream components of the combustor. This requires designing an efficient cooling technique in this area. In the traditional cooling system, the increase in blowing ratio enhances cooling effectiveness. But, the coolant does not well attach on the surface at higher blowing ratios. This necessitates restructuring the cooling holes. A useful way can be trenching cooling holes at the combustor end wall surface and the alignment of row trenched holes; however, this has not been seriously considered as a solution up to the present time. The major effects of cylindrical and row trenched cooling holes with alignment angle of 0, +60 and 90 degrees at blowing ration, BR=3.18 on the effectiveness of film cooling near the combustor end wall surface is a subject to study in detail. In the current study, researchers used a FLUENT package 6.2.26 to simulate a 3-D model of a Pratt and Whitney gas turbine engine. In this research, a RANS model was used to analyses the flow behaviour on the passageways of internal cooling. In the combustor simulator, the dilution jets and cooling flow staggered in the streamwise direction and aligned in the spanwise direction as well. In comparison with the baseline case of cooling holes, the application of row trenched hole near the endwall surface increased the effectiveness of film cooling from 75% to 100% for different trench cases.

Keywords:
Gas Turbine; Film-Cooling; Cylindrical Hole; Trench Hole

1. Introduction

Advanced gas turbine industries are trying for higher engine efficiencies. Brayton cycle is a key to achieve this purpose. In this cycle, to have higher gas turbine engine efficiency, the combustor`s outlet temperature must increase [1]. But such hot flows cause non-uniformities at the end of the combustor and the inlet of the turbine and damage the critical parts. Film cooling is the most well-known method of preservation. In this technique, a low temperature thin layer attaches on a surface and protects it against hot streams. To get better film cooling performance, it is needed to increase

* *Corresponding author.*
E-mail address: ekianpour@gmail.com (Ehsan Kianpour)

blowing ratio. Blowing ratio increment has an intense effect on the heat transfer, particularly in the hole region. According to the importance of this study, a broad literature survey was done to get the Fundamental data.

Harrington *et al.,* [2] presented a simulated flat plate. The research was a computational and experimental one aimed at investigating a full coverage of adiabatic film cooling effectiveness. The focus of the findings was on the effects of ten rows of normal short cooling holes with a length of l/D = 1.0 at large density coolant jets and high mainstream turbulence intensity. The test results indicated that considering the blowing ratio, the maximum adiabatic film cooling effectiveness was attained near the area, which covered four to eight rows of cooling holes. Furthermore, the interaction of injected flows sprayed from the cooling rows limits the maximum effectiveness. On the other hand, film cooling effectiveness on the curved surface was investigated numerically by Koc *et al.,* [3]. They highlighted that the curvature of the surface and the blowing ratio affect the film cooling effectiveness.

Hale *et al.,* [4] measured the effects of geometry parameters such as l/D ratios, injection angles, co-flow and counter-flow plenum feed configurations on surface adiabatic film cooling effectiveness near the cooling holes. The results showed that short injection holes increased the film cooling and developed wide cold region downstream of the cooling holes. Their findings were identical with those of Burd and Simon`s [5], as the relevance of the length of injection hole to effectiveness of film cooling could be well understood. In another study, Azzi and Jurban [6] forecasted the film cooling thermal fields for a simulated cylindrical cooling row with the following ratios: extended range of length to diameter = 1.75/8, and fixed pitch to diameter = 3.0. The inclination angle was 35 degrees and the film hole was 12.7 mm in diameter. Another finding by Bernsdorf *et al.*, [7] showed that in short cooling injection holes, effectiveness of film cooling is related to the injection hole length and angle.

Tarchi *et al.,* [8] investigated the effects of large dilution holes. These holes were placed within the injecting slot and eruption array. The flat plate cross section duct contained 270 cooling holes located in 29 staggered rows. The holes were 1.65 mm in diameter and had a length to diameter ratio of 5.5 and a stream-wise angle of 30 degrees. The dilution hole was 18.75 mm in diameter. It was located at the 14th row of cooling holes. Tarchi *et al.,* [8] measured the local heat transfer coefficient and adiabatic film cooling effectiveness for both conditions, with and without backward facing step, at three different blowing ratios of BR=3.0, BR=5.0 and BR=7.0. Tarchi *et al.*, [8], Milanes *et al.,* [9], Barringer *et al.,* [10], Li *et al.,* [11] and Scrittore [12] illustrated that with using backward step, at downstream the dilution hole the adiabatic film cooling effectiveness reached to $\eta_{aw} = 0.65$.

Vakil and Thole [13] presented experimental results of the study of temperature distribution inside a combustor simulator. In this study, a real large scale of combustor was modelled. This model contained four different cooling panels with many cooling holes. Two rows of dilution jets could be seen in the second and third cooling panels. The first row had three dilution jets and the second one had two jets. While the first and second panels were flat, two other panels angled with an angle of 15.8 degrees. In this study, a real large scale of combustor was simulated and high momentum dilution jets and the coolant flow were injected into the main flow. The results indicated that high temperature gradient was developed upstream of the dilution holes. Kianpour *et al.,* [14,15] re-simulated the Vakil and Thole`s combustor. They offered various geometries of cooling holes. The temperatures near the wall and among the jets were higher for the baseline cooling whereas the central part of the jets was cooler in trench cases. Using a turbine vane cascade, the effects of shallow trenched holes (d=0.5D) were investigated by Somawardhana and Bogard [16] to improve the performance of film cooling. They measured the effectiveness of film cooling under blowing ratios from BR=0.4 to 1.6. The findings indicated that upstream obstructions reduced the effectiveness by 50%. However, downstream obstructions increased the film cooling performance. The film cooling performance was slightly affected by a combination of obstructions near the upstream obstructions. Using a narrow trench, they dramatically modified the cooling performance and reduced the effects of surface roughness decrease. The results agreed with Shupping`s [17] findings. Furthermore, Somawardhana and Bogard [16] showed higher film cooling effectiveness for the trenched holes which resulted from the net heat flux reduction was higher than the baseline case, while the heat transfer coefficient was approximately constant for both cases. In contrast, Yiping and Ekkad [18] showed that trenching cooling holes increased heat transfer coefficient. Ai *et al*., [19] showed the results the ratio of coolant momentum and the cooling effectiveness was reduced while at low blowing ratios the performance became better by traditional cooling holes.

In agreement with the study background, several authors motivate the author to do this research. Endwall of the combustor can be damaged by the hot gases which flow inside a combustor simulator and increasing the film cooling effectiveness above these surfaces is an important issue which attracts less attention till now [20,21]. In addition, most of the studies paid attention on the using of trenched holes at the leading edge of the turbine blades and in most of them the application of these holes at the endwall combustor is not considered. The alignment angle of the trenched cooling holes is a topic which not considered in the past researches. Also, this approach of cooling holes can be utilized by engine designers at the fore side of the turbine vanes. Also in order to measure the validity of the results, a comparison between the data gained from this study and Vakil and Thole [13] and Stitzel and Thole [22] project was carried out.

2. Methodology

In the present study, the combustor simulator applied was a 3-D representation of a Prat and Whitney gas turbine engine. In a real combustor, film cooling jets are used to form a low temperature layer of air that covers the combustor's inner and outer coating. Without effective cooling, this cover is going to melt and cause disaster effects. As seen in Figure 1, the combustor was a three-dimensional container. The width, height and length of this container was 111.8cm, 99.1cm and 156.9cm respectively. The container converged from X/L=0.51 and contraction angle was 15.8 degrees. Inlet and outlet cross-sectional area of the combustor simulator was 1.11m^2 and 0.62m^2. The test section contained two symmetric surfaces on the top and bottom of the combustor but the fluid only flowed through bottom passage. The lengths of the panels were 39cm, 41cm, 37 and 43 centimeters respectively. In addition, the first two panels were flat and have constant sectional area. However, the last two panels were inclined at the contraction angle and yield the exit sectional area of the combustor simulator. The panels were 1.27cm thick, and due to the low thermal conductivity (k=0.037 W/mk), adiabatic surface temperature measurements were possible.

Dilution flow injection is beneficial to decline the temperature of the hot exit gases of the combustor simulator and prevent the detrimental effects of the critical components. Two different dilution rows were considered within the second and third panel of cooling panels. The dilution flow injected into the mainstream flow vertically, while, the dilution hole in the third panel was angled at 15.8degrees from the vertical axis. The first row of dilution jets included three holes and it was placed at 0.67m downstream of the combustor simulator inlet. These holes were 8.5cm in diameter. The second row contained two dilution holes and was located at 0.23m downstream of the first row of dilution holes' center. These holes' diameter was 1.4 times more than the first one at 12.1cm. The centerline of the second row was staggered with respect to those of the first row. In the present research, the combustor simulator contained four arrangements of cooling holes. For the verification of findings, the first arrangement (baseline or case 1) was designed similar to the Vakil and Thole [13] combustor simulator. The length of these cooling holes was 2.5cm and they drilled at an angle of 30deg from the horizontal surface.

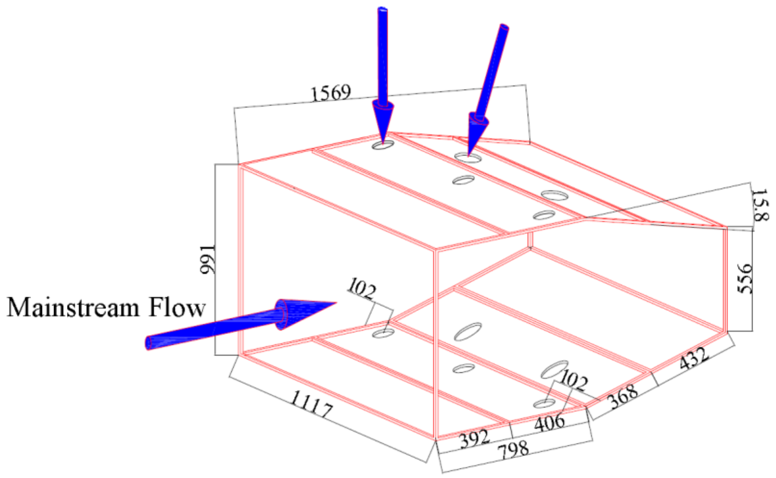

Fig. 1. Schematic view of the combustor simulator

The film-cooling holes were 0.76cm in diameter. Except the baseline case which introduced, to investigate the effects of cooling holes trenching, different rows of trenched holes with alignment angles of 0, 90 and +60 degrees with trench depth and width of 0.75D and 1.0D were considered (Figure 2).

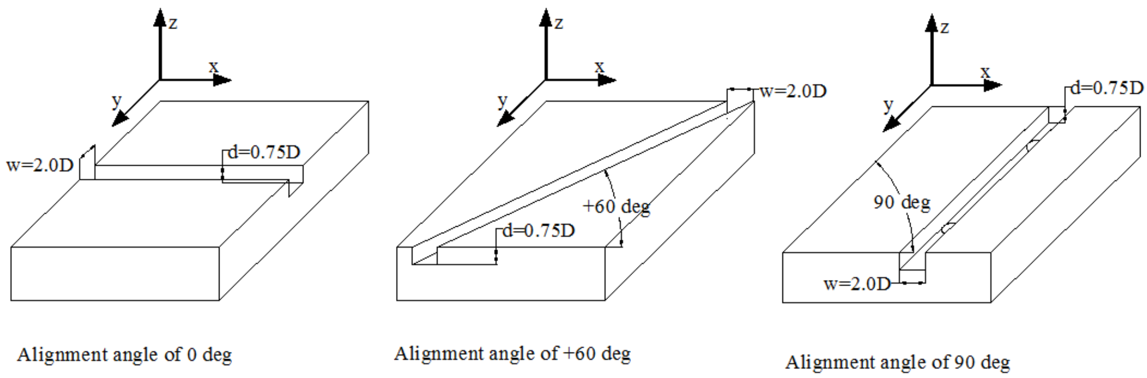

Fig. 2. Arrangements of row trenched cooling holes

Furthermore, the length of cooling holes after trenching declined to 1.36cm. Furthermore, coolant blowing ratio was equal to BR=3.18. For this study, it was so important to define a coordinate system inside a combustor simulator to achieve the same references for all research cases and investigate the 3D combustor simulator. Therefore, the Cartesian coordinate system (x, y and z) was selected. All coordinates were non-dimensionalized by the combustor simulator height (H_{in}), length (L) and width (W) respectively.

Figure 3 shows the observation planes which are used to measure the film-cooling effectiveness distribution for baseline case and three different configurations of row trenched cooling holes. The observation planes of 0p, 1p, 2p, and 3p and 0s were placed in pitchwise and streamwise direction respectively. Plane 0p was located at x=35.1 cm. The distribution of film cooling momentum was computed along this panel. This plane lengthened within half of the combustor in the spanwise direction. The observation plane height extended from z=0cm to z=10cm. Plane 1p was located at the trailing edge of the first row of dilution jet.

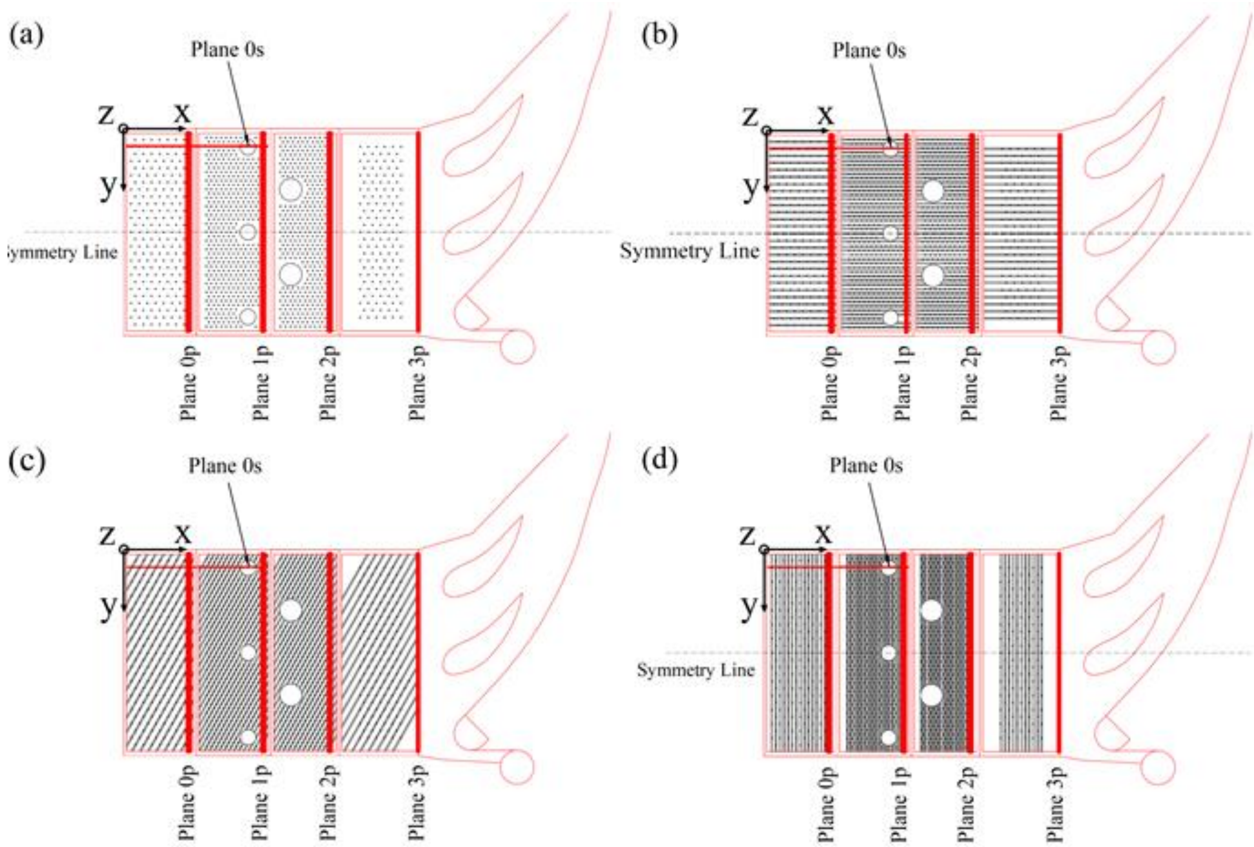

Fig. 3. Location of the observation planes: (a) baseline, (b) case 2 (c) case 3, (d) case 4

The importance of this plane was to identify the stream-wise behavior of the dilution jets first row. About 8×10^6 tetrahedral meshes were selected. This quantity of nodes allowed appropriate convergence for corrected meshing; the thermal and flow characters would have the similar variation as the higher refinement mesh. According to the considered blowing ratio at the inlet of control volume, the boundary condition of inlet mass flow was considered at the inlet. to limit the interaction region between fluid and combustor wall, slip-less boundary condition and wall boundary condition were considered. In addition, two different boundary conditions of uniform flow and pressure outlet was selected at the inlet and outlet of combustor respectively. Totally, according to the symmetries of the Pratt and Whitney gas turbine engine combustor, symmetry boundary condition was used. Gambit package was selected to mesh the combustor simulator and the model was analyzed by Fluent 6.2.26 software.

In the case of the investigated meshes of the model, the combustor simulator domain has an increased mesh resolution due to the rectangular combustor being exposed to the complex flow effects. The mesh is refined in the grids from 1×10^6 to 8×10^6 where 7×10^6 and 8×10^6 represent coarse, medium, and fine mesh generated for the standard k-ε turbulence model. The average temperature increased from 314 to 321 as shown in Figure 4.

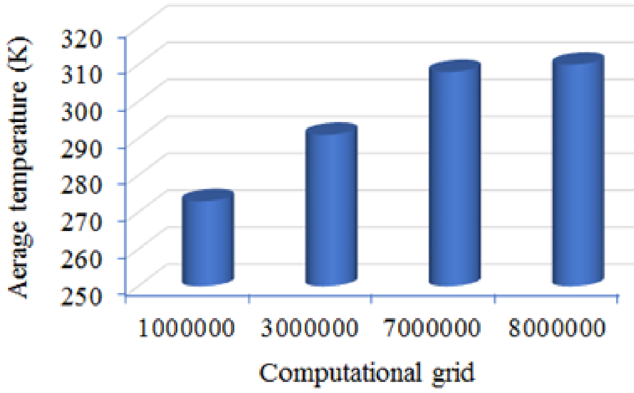

Fig. 4. The average temperature of all the investigated meshes in mesh independency study

The near-wall flow fields require special treatment. In RANS turbulent model, there are two main approaches for the determination of the shear-stress at a wall: Wall Function approach, which determine the shear-stress at the wall from semi-empirical equations, and Near Wall Model Approach. The y+ is the dimensionless quantity for the distance from the wall up to the center of the first grid cell. Accurate presentation of the flow in the near-wall region determines successful prediction of wall-bounded turbulent flows. Values of In the Wall Function Approach method y^+ are most desirable for wall functions (Figure 5).

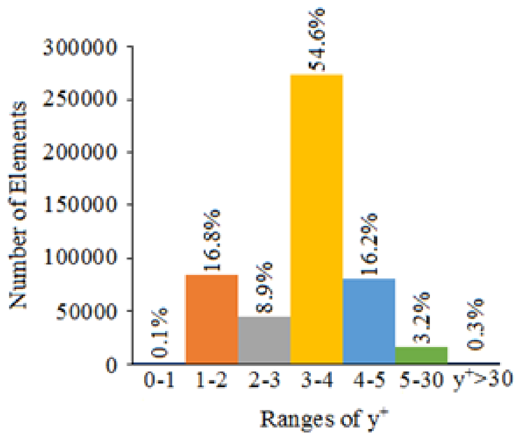

Fig. 5. Corresponding wall y+

It then makes a comparison in regard to the thermal field between numerical studies done by the finite difference method and the computational analyses previously performed by Vakil and Thole [13] and Stitzel and Thole [20]. The numerical method considered a transient, incompressible turbulent flow by means of the k–ε turbulent model of the Navier–Stokes equations expressed as follows:

Continuity equation

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}=(\rho u_i u_j)=-\frac{\partial P}{\partial x_i}+\frac{\partial \tau_{ij}}{\partial x_i}+\rho g_i+\vec{F} \quad (1)$$

Momentum equation

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}\frac{dx}{dt}+\frac{\partial}{\partial y}\frac{dy}{dt}+\frac{\partial}{\partial z}\frac{dz}{dt}=-\rho(\nabla.V) \tag{2}$$

Energy equation

$$\frac{\partial}{\partial t}(\rho E)+\frac{\partial}{\partial x_i}\left(u_i(\rho E+P)\right)=\frac{\partial}{\partial x_i}\left(K_{eff}\frac{\partial T}{\partial x_i}-\sum_j h_j j_j+u_j(\tau_{ij})_{erf}\right)+S_h \tag{3}$$

and k–ε equation

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k-\rho\varepsilon \tag{4}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}P_k-C_{1\varepsilon}^2\rho\frac{\varepsilon^2}{k} \tag{5}$$

The convective and diffusion terms were approximated by the first-order upwind and central differencing scheme. To investigate the convergence limit, the control volume mass residue has been estimated and the maximum value has been used. For this research, the criterion of convergence has chosen 10^{-4} . The following equation is to determine the effectiveness of film-cooling.

$$\eta=\frac{T-T_\infty}{T_C-T_\infty} \tag{6}$$

Here, T is the local temperature, and T_∞ and T_C is the temperature of the mainstream and coolant.

3. Results

The findings of the current research were compared with the experimental collected results which was done by Vakil and Thole [13] and numerical findings gathered by Stitzel and Thole [20]. The effectiveness of film-cooling was compared in plane 1p and 2p at y/W=0.4. The deviations between the results of current research and benchmarks were computed by the following equation.

$$\%Diff=\frac{\sum_{i=1}^{n}\frac{x_i-x_{i,benchmark}}{x_{i,benchmark}}}{n}\times 100 \tag{7}$$

The deviation was equal to 9.76% and 8.34% compared to Vakil and Thole [13] measurements and Stitzel and Thole [20] estimation for plane 1p and 13.36% and 11.96% in comparison with study done by Shuib *et al.,* [20] and Vakil *et al.,* [13] for plane 2p (Figure 6).

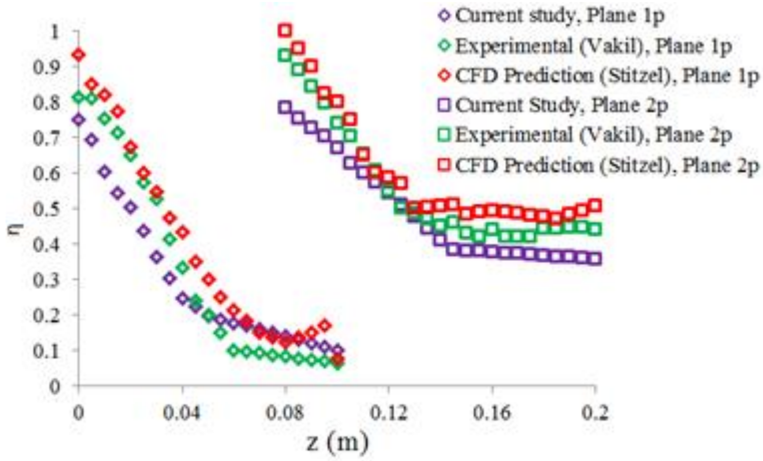

Fig. 6. The film cooling effectiveness comparison of planes 1p and 2p along y/W=0.4

The row trenched holes with alignment angles of 0 and ±60 degrees' effects are seen in Figure 7. In these cases, cooling blowing is increasing and the injection of dilution jets is constant. The results showed that cooling effectiveness was awhile higher near the combustor endwall in trench cases especially for row trenched holes with alignment angles of 90 and +60 degrees compared to the baseline. But, no significant improvement was seen along the combustor endwall. The thermal field contours may show that film-cooling is entrained by the upward motion of the dilution jet merely at the downstream of the dilution jet and near the corners. Besides, the temperature was not much higher ($0<\eta<0.05$) for the trenched holes with an alignment angles of 90 and +60 degrees at a position of 18cm<y<28cm. Of course, the hot region extended from y=38cm to y=50cm.

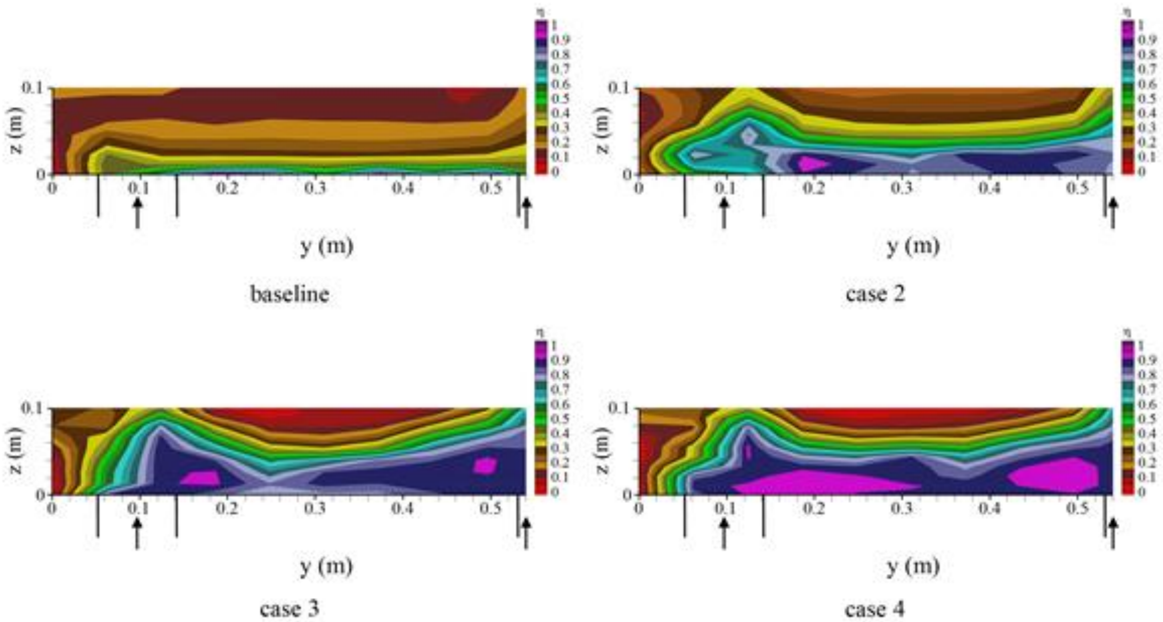

Fig. 7. Film-cooling effectiveness in plane 1p

The effectiveness of film cooling distribution in observation plane 2p at blowing ratio of 3.18 is shown in Figure 8(a-d). The most important difference between these schemes is the coolant injection into the main flow. Exactly downstream the second row of dilution jets, more effective coolant layer (10cm<y<50cm) was formed by the trenched holes than the baseline case on the critical surfaces downstream the combustor simulator. The significant protective layer (0.9<η<1.0) was seen for the row trenched holes with alignment angles of 0 and +60 and 90 degrees. While, for the baseline, the most effective layer was created up to 8% of the combustor inlet height, for three other cases, this layer reached at z=18cm for the trenched holes with alignment angles of +60 and 90 degrees. For the trenched holes with alignment angle of 0 degree, the maximum height of the layer was 20 cm, equal to 20% of the combustor inlet height. This gave the highest cooling layer among all the cases. Hence, a warmer area (0.15<η<0.20) expanded at the right (0cm<y<4cm) and left (48cm<y<52cm) sides of the thermal field contour.

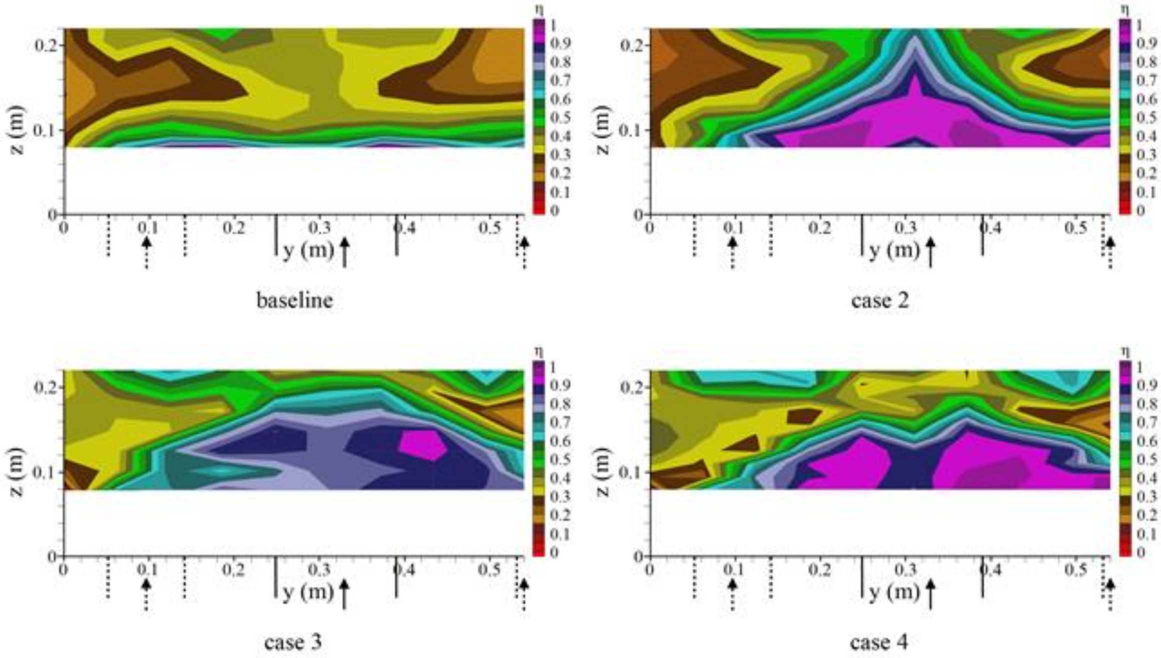

Fig. 8. Film-cooling effectiveness of plane 2p

The velocity vectors *v* and *w* superimposed on the film cooling effectiveness contour in plane 0s can be seen in Figure 9. The coolest core of dilution jet can be seen between 64.3cm<x<70.6cm as it enters into the mainstream. Just downstream of the dilution jet, the layer of low temperature region reached to z=7 cm which is 7% of the combustor inlet height for the trenched holes with the alignment angle of 90 degrees, after passing through the first row of dilution, the overall bulk reduction at the trailing edge of the first row of dilution jet (1D1) temperature of η=0.25 was calculated. However, the high temperature region downstream of the first row of dilution jet reached to η=0.05 for the trenched holes with the alignment angle of 60 degrees. The minimum temperature region η=0.6 in this area was created by applying trenched holes with the alignment angle of 90 degrees. Interesting was the film cooling interaction with the dilution jet itself: At the leading edge of the dilution jet, a relatively thick layer of film-coolant stagnated onto the dilution jet and carried away by the dilution jet. A blockage to the flow was naturally created so that the stagnated flow would be left downstream of the jet when a dilution jet was injected into a cross flow. The shear forces at the trailing edge of the jet took advantage of the stagnated flow by entraining it, and consequently created the entrainment or recirculating region behind the dilution jet. At the trailing edge of the jet, thermal contours could be seen to spread as a result of turbulent mixing induced by the dilution flow. Furthermore, the entrainment of the film-cooling at the trailing edge of the dilution hole was clearly visible around the centreline of the dilution jet. However, the entrainment effects would become weaker with the increase of distance from the core of the jet, especially in the case of the trenched holes with the alignment angles of 90 and +60 degrees. The research findings confirm the fact that as the distance from the jet centreline increased, the non-mixed-out, cool cores of the numerous film-cooling jets became increasingly visible. On neither side of the dilution jet (between z=7cm to z=10cm), did not combustor gases mix with the dilution jet or the film cooling.

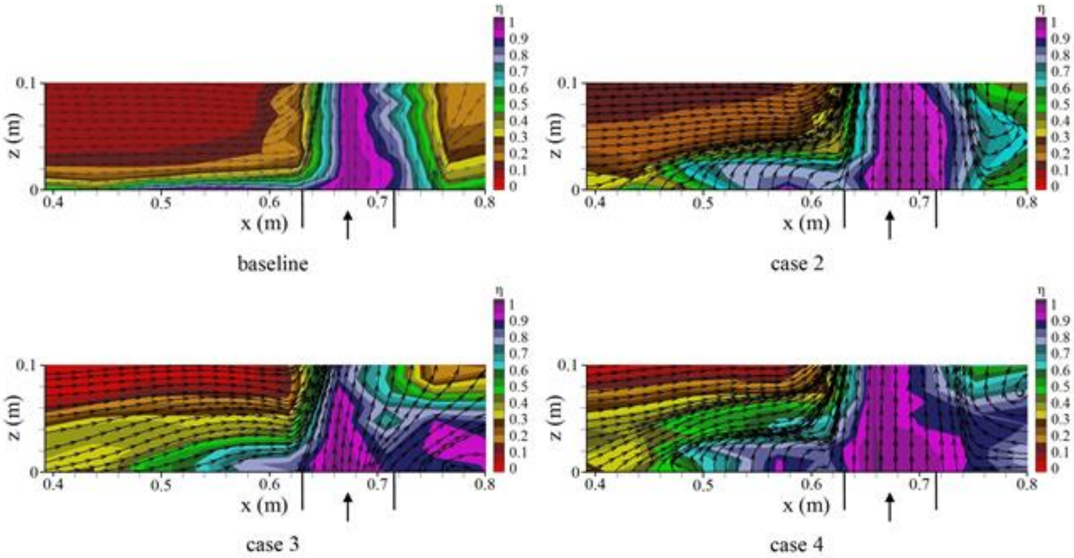

Fig. 9. Film-cooling effectiveness in plane 0s

Figure 10 shows the streamwise film cooling distribution along a first row of dilution jet for all configurations at M=3.18.

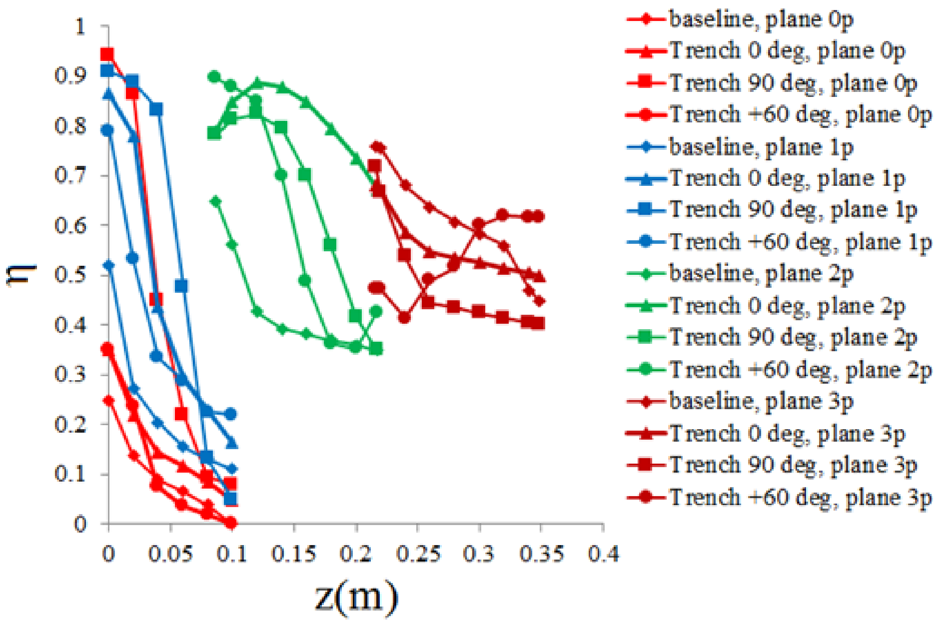

Fig. 10. Film cooling effectiveness distribution

The trenched holes with alignment angles of 0 and 90 degrees performed the most effectively. This was 1.75 times as much the film cooling effectiveness of the baseline case. The film exiting the trenches created a new boundary and enhanced heat transfer coefficients immediately downstream of the trench. The baseline was the lowest among all cases. In plane 2p, the row trenched holes with alignment angles of 0 and 90 degrees indicated almost similar effectiveness levels (η=0.785). The baseline had a 13% reduction in cooling performance with an increase in its blowing ratio. For plane 3p, the results showed that the highest film cooling effectiveness enhancement with blowing ratio increase was produced by baseline case (58%) and the trenched holes with alignment angle of +60 degrees (38.8%).

4. Conclusions

In the current research, a numerical study was done to get a better understanding about effectiveness of film cooling and the row trenched and cylindrical holes' effects at the combustor outlet. The combustor combines the interaction of two rows of dilution jets, which are staggered in the stream-wise direction and aligned in the span-wise direction, with that of film-cooling along the combustor liner walls. In the research, three-dimensional representation of a Pratt and Whitney rectangular gas turbine engine was modelled and analysed by FLUENT software package 6.2.26 and RNG k-ε turbulence model because they are widely used in the industries and universities. Trenched cooling hole increases film effectiveness, however, the coolant, in effect, stays closer to the surface and does not allow main entrainment and also spreads laterally. It also provided significant lateral spreading and stronger coverage. Trenched holes, adjacent to end wall surfaces, perform more efficiently. It appears that an effective film cooling can be made by row trenched cooling holes with alignment angle of +60 degrees for the observation planes of 0p and 2p. However, for plane 1p, i.e. at the trailing edge of second cooling panel, the row trenched holes with alignment angle of +60 degrees increased film cooling effectiveness by 75% which is much higher than other configurations. Concerning the jet advancing downstream, different cooling holes' configurations showed the same behaviour at the combustor exit. The findings of film cooling effectiveness observed in the current study and the experimental data and the CFD prediction collected by Vakil and Thole [16] and Stitzel and Thole [22] indicate the presence of similarities and differences. The predicted thermal field data indicated an under-predicted measurement for the current study in contrary to experimental results. Based on the results of the current study, the following suggestions are offered for further research. It was not quite clear which trenched holes with different alignment angles have stronger effects on the film cooling effectiveness; because the behaviour of the holes was completely different at low and high blowing ratios and even from one observation plane to another one and therefore it is highly recommended to see the effects of a combination of row trenched cooling holes.

Acknowledgement
The authors would like to thank Universiti Teknologi Malaysia and the Ministry of Education of Malaysia for supporting this research activity. This research was financially supported by the Research University Grant 06H23.

References
[1] Kianpour, Ehsan, Nor Azwadi Che Sidik, and Iman Golshokouh. "Film cooling effectiveness in a gas turbine engine: a review." *Jurnal Teknologi* 71, no. 2 (2014): 25-35.
[2] Harrington, Mark K., Marcus A. McWaters, David G. Bogard, Christopher A. Lemmon, and Karen A. Thole. "Full-coverage film cooling with short normal injection holes." *Journal of Turbomachinery* 123, no. 4 (2001): 798-805.
[3] Koc, Ibrahim, Yasar Islamoglu, and Unal Akdag. "Investigation of film cooling effectiveness and heat transfer coefficient for rectangular holes with two rows." *Aircraft Engineering and Aerospace Technology 81*, no. 2 (2009): 106-117.
[4] Hale, C. A., Michael W. Plesniak, and Satis Ramadhyani. "Film Cooling Effectiveness for Short Film Cooling Holes Fed by a Narrow Plenum." *Journal of Turbomachinery* 122, no. 3 (2000): 553-557.
[5] Burd, Steven W., and Terrence W. Simon. "Measurements of Discharge Coefficients in Film Cooling." *Journal of Turbomachinery* 121, no. 2 (1999): 243-248.

[6] Azzi, Abbes, and Bassam A. Jubran. "P Numerical Modeling of Film Cooling from Short Length Stream-Wise Injection Holes." *International Journal of Heat and Mass Transfer* 39, no. 4 (2003): 345–353.
[7] Bernsdorf, Stefan, Martin G. Rose, and Reza S. Abhari. "Modeling of Film Cooling—Part I: Experimental Study of Flow Structure." *Journal of Turbomachinery* 128, no. 1 (2006): 141-149.
[8] Tarchi, Lorenzo, Bruno Facchini, Francesco Maiuolo, and Daniele Coutandin. "Experimental investigation on the effects of a large recirculating area on the performance of an effusion cooled combustor liner." *Journal of Engineering for Gas Turbines and Power* 134, no. 4 (2012): 041505.
[9] Milanes, David W., Daniel R. Kirk, Krzysztof J. Fidkowski, and Ian A. Waitz. "Gas Turbine Engine Durability Impacts of High Fuel-Air Ratio Combustors: Near Wall Reaction Effects on film-Cooled Backward-Facing Step Heat Transfer." *Journal of Engineering for Gas Turbines and Power* 128, no. 2 (2006): 318-325.
[10] Barringer, Michael D., O. T. Richard, J. P. Walter, Sarah M. Stitzel, and Karen A. Thole. "Flow Field Simulations of a Gas Turbine Combustor." *Journal of Turbomachinery* 124, no. 3 (2002): 508-516.
[11] Li, L., T. Liu, and X. F. Peng. "Flow characteristics in an annular burner with fully film cooling." *Applied thermal engineering* 25, no. 17-18 (2005): 3013-3024.
[12] Scrittore, Joseph J., K. A. Thole, Steven W. Burd. "Investigation of Velocity Profiles for Effusion Cooling of a Combustor Liner." *Journal of Engineering for Gas Turbines and Power* 129, no. 3 (2006): 518-526.
[13] Vakil, Sachin S., and K. A. Thole. "Flow and Thermal Field Measurements in a Combustor Simulator Relevant to a Gas Turbine Aero engine." *Journal of Engineering for Gas Turbines and Power* 127, no. 2 (2005): 257-267.
[14] Kianpour, Ehsan, Nor Awadi C. Sidik, and Mazlan Abdul Wahid. "Cylindrical and Row Trenched Cooling Holes with Alignment Angle of 90 Degree at Different Blowing Ratios." *CFD Letters* 5, no. 4 (2013): 165-173.
[15] Kianpour, Ehsan, Nor Azwadi Che Sidik, and Iman Golshokouh. "Measurement of film effectiveness for cylindrical and row trenched cooling holes at different blowing ratios." *Numerical Heat Transfer, Part A: Applications* 66, no. 10 (2014): 1154-1171.
[16] Somawardhana, Ruwan P., and David G. Bogard. "Effects of obstructions and surface roughness on film cooling effectiveness with and without a transverse trench." *Journal of Turbomachinery* 131, no. 1 (2009): 011010.
[17] Shuping, Chen. " Film cooling enhancement with surface restructure." PhD theses., University of Pittsburgh, 2008.
[18] Lu, Yiping, and Srinath Ekkad. "Predictions of film cooling from cylindrical holes embedded in trenches." In *9th AIAA/ASME Joint Thermophysics and Heat Transfer Conference*, p. 3401. 2006.
[19] Ai, Weiguo, Robert G. Laycock, Devin S. Rappleye, Thomas H. Fletcher, and Jeffrey P. Bons. "Effect of particle size and trench configuration on deposition from fine coal flyash near film cooling holes." *Energy & Fuels* 25, no. 3 (2011): 1066–1076.
[20] Shuib, W., Idris, A.C., Saad, M.R., Hasan, M.F., Rahman, M.R.A. "Study on the effect of bluff body with slit in the micro- channel combustor." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 52, no. 2 (2018): 215-222.
[21] Hoseini, S.S., Najafi, G., Ghobadian, B. "Thermal and fluid simulation of a new diesel engine cooling exhaust gas recirculation system to reduce exhaust gas emissions." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 51, no. 2 (2018): 197-208.
[22] Stitzel, Sarah, and Karen A. Thole. "Flow field computations of combustor-turbine interactions relevant to a gas turbine engine." In *ASME Turbo Expo 2003, collocated with the 2003 International Joint Power Generation Conference*, pp. 175-183. American Society of Mechanical Engineers, 2003.