VOL. 10, NO. 6, APRIL 2015 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

NUMERICAL SIMULATION OF FILM COOLING OVER FLAT PLATE

Ali S. Bahr Ennil[1] and Abdulhafid M. Elfaghi[2]
[1]Department of Aeronautical Engineering, Engineering Academey Tajoura, Libya
[2]Department of Aeronautical Engineering, University of Zawia, Libya
E-Mail: hafied@zu.edu.ly

ABSTRACT

The effect of film cooling over flat plate is investigated using the commercial CD code; Fluent 6.3. The computational domain includes the coolant supply tube as well as the main mixing region. A tube L/D of 4 and injection angles of (30°, 60°, and 90°) were employed for blowing ratio of (0.33, 0.5, and 1.67), and a density ratio of 1.14. Adiabatic film cooling effectiveness distributions were also determined for inline and staggered arrangements. The main observation from this study that the 30° hole gave larger effectiveness values than 60° and 90° at the blowing ratio of 0.33 with the same length-to-diameter ratio. The maximum effectiveness was achieved with a blowing ratio of 0.5. The results show that the increase of blowing ratio negatively affects film cooling, such that for the blowing ratio of 1.67 the injected coolant tends to lift off from the wall due to the increase of the wall normal momentum. The comparisons for numerical results with experimental data are presented.

Keywords: film cooling, adiabatic effectiveness. injection angle, blowing ratio.

INTRODUCTION

The advance of turbine engine technology has led to higher turbine inlet temperatures. This requires active cooling in order to maintain the blades of the turbine engine at a safe temperature, since the excessive high temperature levels will reduce the life of blades and can even cause the failure of those blades. Film cooling is a widely used technique to achieve this goal. Due to the large number of influencing parameters, design optimization through experiments can be quite expensive and time consuming.

Consequently a reliable computationally efficient predictive procedure would be extremely beneficial. Even for a row of holes, the flow field is quite complex with a wide variety of influences parameters such as hole injection angle, hole diameter and shape, blowing ratio, lateral spacing, and the effect of the length of the coolant supply tube.

The present study used FLUENT 6.3 to solve the flow field and properties through the different configurations with standard k-ε model. Temperature contours, average effectiveness values and effectiveness contours has been presented. The obtained numerical results of this study are compared with experimental data.

Numerical model

The flat plate to be film is illustrated in Figure-1. Table-1 shows test cases data.
The geometry of numerical model is as follows:

- The plate is 360mm wide ×1000mm long ×163mm thick.
- Hole diameter (D) is 10mm.
- Pitch to diameter is 3.
- Injection angles $30^{0,}$ 60^0, and 90^0.

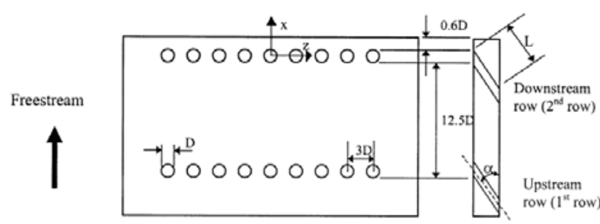

Figure-1. Flat plate configuration.

Table-1. Run cases data.

Injection angle	Configurations	P/D	s/D	Blowing ratio
30 , 60, and 90	One hole		12.5	0.33, 0.5, 1.0, 1.67
	Two inline rows	3	12.5	
	Two staggered row	3	12.5	

The following assumptions are made:

- Both the hot mainstream gas and the cooling fluid are approximated as air.
- The flow field is symmetrical about the vertical plane passing through the stream wise centerline of a cooling hole (hole-center plane).
- The flow field is symmetrical about the vertical plane passing in the mid-distance between cooling holes (mid-pitch plane).

Mesh generation

The mesh is generated using GAMBIT®2.3.16. It is crucial that the grid size be as small as possible at boundaries (near flat plate and cooling hole internal walls). It is also important that the grid volumes be as large as possible to reduce the computational time and allocated memory. The grid was non-orthogonal in the tube and near the injection hole. Conservation was maintained by employing control volume concepts similar to those employed with unstructured meshes. The size function applied has a minimum grid dimension of 0.5 mm at the tested flat plate and cooling hole. It uses a growth rate of 1.05 and maximum grid dimension of 1mm. A zoomed snapshot of the mesh control volume is shown in Figure-2 in more detail.

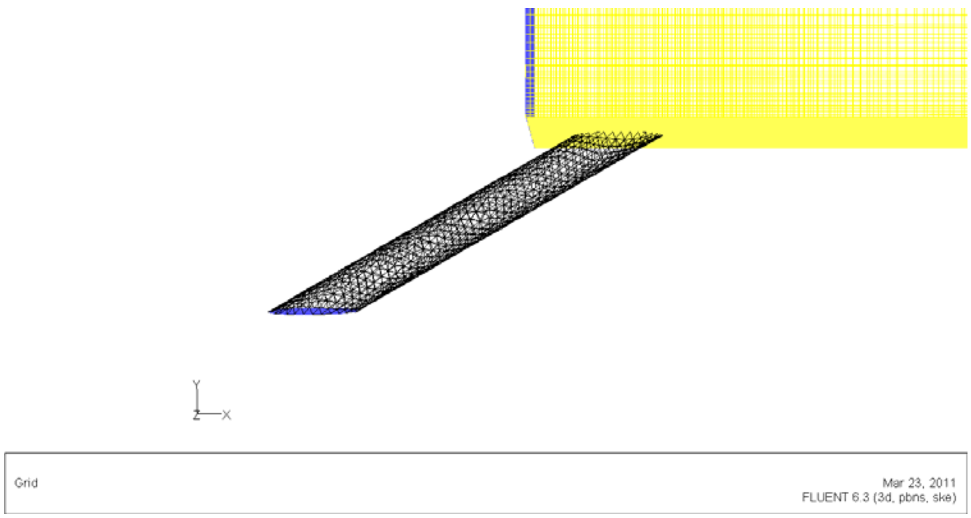

Figure-2. The mesh generation.

Boundary conditions

The flow field and temperatures in numerical models studied are solved using FLUENT6.3, for the following boundary conditions:

a) **Hot main stream inlet** `

The mainstream gases are approximated as air, with a temperature 333 K. The inlet mainstream flow is considered to have a uniform velocity of 10 m/s. As measured by [14], the turbulence level is 11% and turbulent length scale of 5 mm.

b) **Coolant inlet**

The coolant inlet is taken to be air, with a temperature 293 K. The inlet coolant flow is considered to have a uniform velocity. Due to the temperature ratio of 333/293, density ratio is maintained at 1.14.

The flat plate surfaces upstream and downstream of the hole injection are considered adiabatic (zero heat flux).

c) **Coolant pipe walls**

The internal walls of the coolant pipe are considered adiabatic.

Periodic boundary conditions

Symmetry boundary condition is applied to the upper boundary of the control volume.

Convergence criterion

Continuity equation, linear momentum equations and turbulence model (k-ε) equations are solved for the mesh volumes to a residual of 10^{-5}, while the energy equation is solved to a residual of 10^{-6}. The different equations are solved implicitly, for seven run cases, for steady state solution using FLUENT6.3.

Grid sensitivity study

Three models are built, for cooling holes (30, 60, and 90 degree) with the same dimensions illustrated in [5], with three different mesh resolutions. These three models contain different numbers of mesh elements, namely 217998, 598830 and 1253317 elements. Studying the effect on centerline effectiveness of film cooling, the model sensitivity to resolution begins to vanish at the third resolution (1253517 cells) with a maximum error of 5%. To minimize the processing time and allocated memory, this resolution is taken in all configurations of the present studies.

RESULTS

The computational methodology implemented in the present research was developed and validated for studying jet- cross flow interactions as described by Yuen *et al*. [5]. CFD analysis was performed using fluent solver and turbulence closure was attained using standard k–ε turbulence model in conjunction with the standard wall functions. The computational results of centerline effectiveness are presented and compared with

ARPN Journal of Engineering and Applied Sciences

experimental results. The results are classified into the following:

Effects of hole angles

In this section, the effect of hole injection angle was studied for three angles (30°, 60°, and 90°) and blowing ratio of 0.33. Figure-3 shows the contours of effectiveness of single hole of 30° holes and a length -to-diameter ratio of 4 with blowing ratio of 0.33. Also the temperature distribution contours is shown in Figure-4. It should be noted that values on effectiveness contours can be drawn in the reverse manner to those on temperature contours. That is; a temperature of 333 K corresponds to zero effectiveness and a temperature of 293 K corresponds to an effectiveness of 100%. When plotting contours of temperatures and effectiveness on RGB color scales, a red color on a temperature map will correspond to a blue color on an effectiveness map. The results show the temperature distribution on the plate, a slight asymmetry is observed around and near the hole.

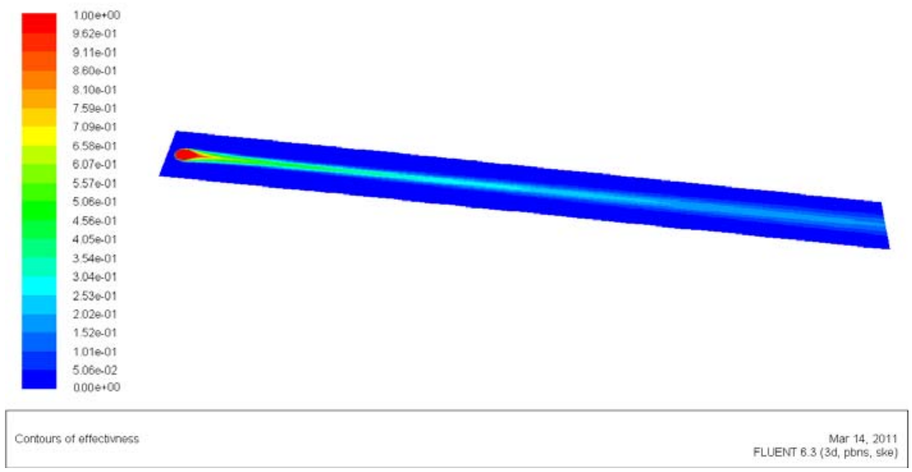

Figure-3. The contours of effectiveness of single hole of 30°.

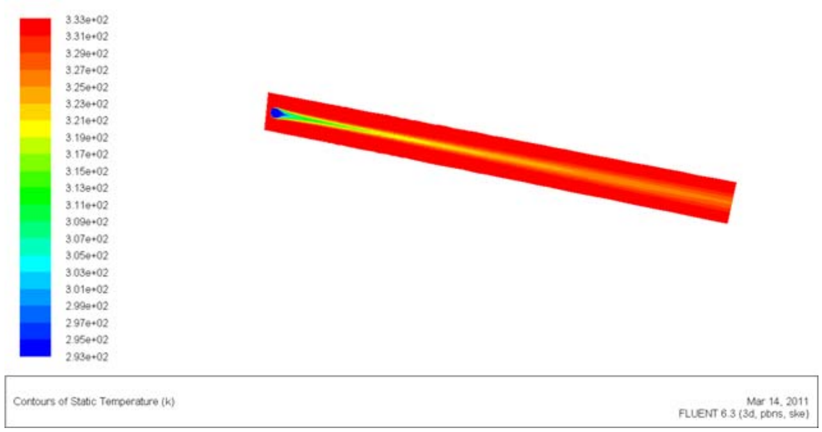

Figure-4. The contours of temperature with of single hole of 30°.

Figures 5, shows the comparison of centerline effectiveness for different injection angles. It is obvious that the injection angle affects the cooling effectiveness and there is an optimum injection angle to achieve high cooling efficiency. However, this angle is correlated with blowing ratio and cooling holes arrangement. As can be from simulation results, the injection angle of 30° provides highest effectiveness for different holes arrangement. At X/D≥ 30 the simulation and experimental results are closely approach, because the wall temperature in this region becomes higher and closer to hot stream temperature. The high free stream turbulence here has significantly shortened the distance that the cooling air travels downstream. This is attributed to the increased mixing of coolant with main flow due to the high turbulence. The high effectiveness regions are narrow as the injection angle increases; however, the effectiveness distribution in the span wise direction becomes extremely uniform. The main observation from this study that the 30° hole gave larger effectiveness values than angles at the blowing ratio of 0.33 with the same length-to-diameter ratio.

ARPN Journal of Engineering and Applied Sciences

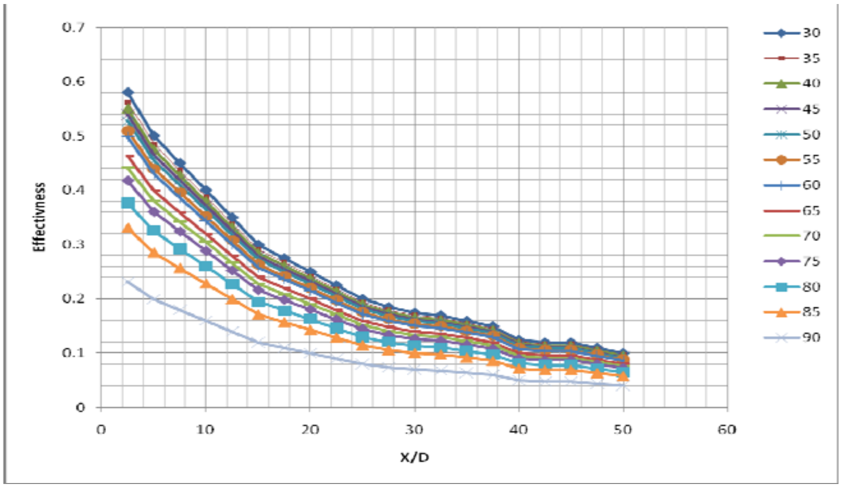

Figure-5. Adiabatic effectiveness for different injection angles.

Effects of blowing ratio

Effect of blowing ratio on effectiveness is studied, by varying the blowing ratio (0.33, 0.5, 1.0, and 1.67) in simulation for single hole with injection angle of 30°. The results are shown in Figure-6. This result shows an effectiveness of around 0.57 at mid-span with a blowing ratio of 0.5 in the immediate and near field regions. This is larger than that of blowing ratios 0.33, 1.0, and 1.67. This provides support for rapid mixing of jet fluid with the flow between the jets. This effect raises the question of turbulence intensity. Also at M=1 a smaller amount of film is spread along the simulated surface.

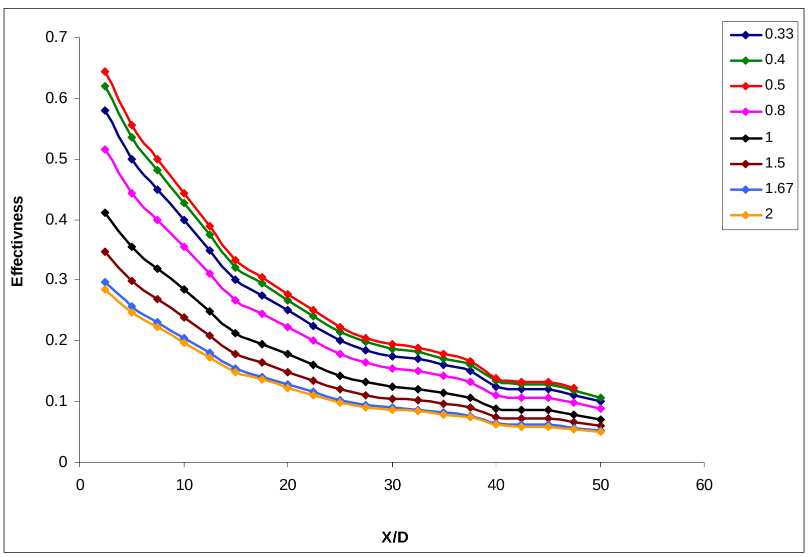

Figure-6. Adiabatic effectiveness for different blowing ratios.

The maximum effectiveness was achieved with a blowing ratio of 0.5, and the results shows that the increase of blowing ratio negatively affects film cooling, where increasing blowing ratio dramatically reduces both the area of coverage of the film cooling air and the magnitude of the effectiveness. This is attributed to the jet lifting off the surface due to higher y-momentum in the jet and stronger kidney vortices. This phenomenon is known with the lift-off effect, shown in Figure-7, which comes as a result of the increased mixing between coolant jet and mainstream. Coolant coverage on the surface is reduced as the blowing ratio increases. So the optimum blowing ratio depends on hole injection angle. At (X/D=30) the blowing ratio has much less effect on effectiveness. The high effectiveness level results from the attachment of the injectant to the wall.

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

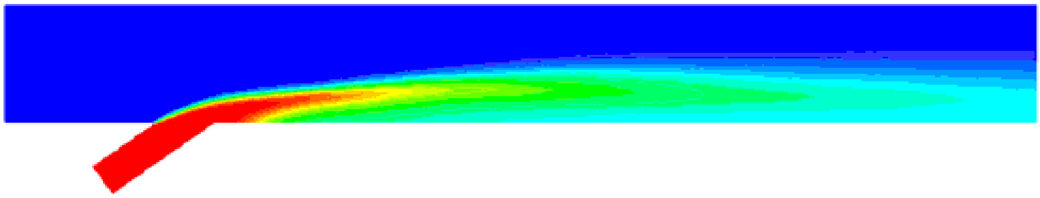

Figure-7. The lift-off phenomena.

Effect of holes arrangement

In this section the numerical study focused on the holes arrangement by comparing two different arrangements in line and staggered. Figure-8 shows the contours of effectiveness of inline holes of 30º injection angle and a pitch-to-diameter ratio of 3 with blowing ratio of 0.33.

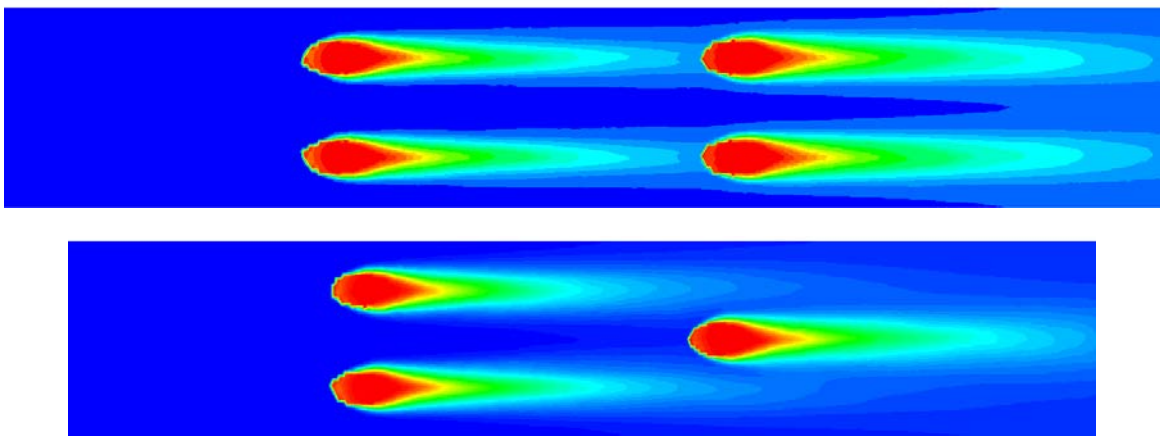

Figure-8. Holes arrangement.

CONCLUSIONS

- The value of effectiveness and sensitivity is affected by many factors as injection angle, blowing ratio, holes arrangement, etc. that makes film cooling performance study not easy.
- This study indicates that the best hole angle is 30^0 at blowing ratio of 0.33, where $\eta_{30^o} \rangle \eta_{60^o} \rangle \eta_{90^0}$.
- The optimum blowing ratio for injection angle 30^0 is 0.5.
- Staggered holes were similar to those inline rows of the same injection angle, and the effectiveness was decreased.

REFERENCES

[1] Kohli A., Bogard D.G. 1997. Adiabatic effectiveness, thermal fields, and velocity fields for film cooling with large angle injection, ASME J. Turbo mach. 119: 352-358.

[2] Walters D.K., Leylek J.H. 1997. A Detailed Analysis of Film Cooling Physics Part I: Stream wise Injection with Cylindrical Holes. ASME Paper 97-GT-269.

[3] Kohli A., Bogard D.G. 1998. Effects of very high free-stream turbulence on the jet-mainstream interaction in a film cooling flow. ASME J. Turbo mach. 120, 785-790.

[4] Guo T, Li S. 2006. Numerical simulation of turbulent jets with lateral injection into a cross flow. J HydrodynSer B. 18(3): 319-23.

[5] C.H.N. Yuen and R.F. Martinez-Botas. Film cooling characteristics of rows of round holes at various stream wise angles in a cross flow: Part1.