A Numerical Study on Heat Transfer Characteristics of Two-Dimensional Film Cooling

Vashista G. Ademane, Vijaykumar Hindasageri and Ravikiran Kadoli

Abstract Determination of reference temperature and heat transfer coefficient in case of three temperature problems such as film cooling is one of the fundamental tasks in the design of gas turbines. In the present work, a two-dimensional numerical simulation is carried out for flat surface with 35° angle of injection from slot in case of film cooling problem. The reference temperature, which is represented as film cooling effectiveness, and heat transfer coefficient on the flat surface for different blowing ratio are studied. Heat transfer coefficient obtained from the present simulation is compared with the experimental results from the literature and found to be matching at lower blowing ratios. Turbulence intensity is found to a major contributor in enhancing the heat transfer coefficient. There is an increase in heat transfer with the blowing ratio due to increased turbulence intensity is observed.

Keywords Film cooling · Effectiveness · Heat transfer coefficient
Turbulence intensity

1 Introduction

The efficiency of gas turbine engines mainly depends upon the temperature of the inlet hot gas. But there is a limitation on the inlet temperature due to the thermal stresses developed in turbine blades. So blades are cooled by taking a part of compressed air and passing them from inside of the blade surface and ejecting out through small holes into the mainstream. The coolant air coming out from the blade surface will

---

V. G. Ademane · R. Kadoli
Department of Mechanical Engineering, National Institute of Technology, Surathkal 575025, Karnataka, India

V. Hindasageri (✉)
Department of Mechanical Engineering, K.L.S's Vishwanathrao Deshpande Rural Institute of Technology, Haliyal 581329, Karnataka, India
e-mail: vkhnitk@gmail.com

D. Srinivasacharya and K. S. Reddy (eds.), *Numerical Heat Transfer and Fluid Flow*, Lecture Notes in Mechanical Engineering,
https://doi.org/10.1007/978-981-13-1903-7_70

create a layer of low-temperature fluid which is a well-established technique known as film cooling.

Major parameters which affect the performance of film cooling are blowing ratio, density ratio, injection angle, hole geometry, turbulence intensity, and mainstream Reynolds number. These parameters are studied on flat surfaces with jet injecting at certain angles to the surface. The experimental work on flat surface film cooling was done by many researchers. Experiments on film cooling have been conducted by [1] from circular holes and later [2] reported heat transfer study. Effect of boundary layer thickness, Reynolds number and free stream turbulence intensity on film cooling is reported by [3, 4] conducted experiment and numerical study.

With the development of different computational techniques and turbulence models, the effort involved in the analysis of the film cooling has reduced. A three-dimensional numerical studies on film cooling was conducted by [5].

Studies are reported on slot jet film cooling by [6, 7], where the secondary air was injected at different angles through a rectangular slot on a flat surface. A 2D numerical simulation of film cooling was carried out by [8, 9]. Numerical and experimental work with various slot angles was performed by [10] and they found that for jet angle larger than 40°, the formation of a recirculation bubble in the downstream of jet. They concluded that the optimum value for the injection angle lies between 30° and 40° to the mainstream. Recently the study of [8] was extended by [11] and conducted numerical investigation for two different Reynolds number with density ratio varying from 1.1 to 5 and blowing ratios of 1–3. They suggested a relation that yields an optimum film cooling effectiveness based on velocity ratio which is nearly equal to sine of the angle of injection.

Even though there are numerous work in the area of film cooling still there is a lack of fundamental understanding on the physics of the fluid behaviour. Many researchers reported on film cooling effectiveness but a few study have been conducted on the heat transfer between the fluid and the surface.

In the present work, a two-dimensional numerical study on a flat surface film cooling is conducted using a commercial simulation software, ANSYS FLUENT. The film cooling effectiveness and the heat transfer coefficient is computed for different blowing ratios. Heat transfer coefficient is compared with the experimental results available in the literature. Effect of turbulence level on the variation of heat transfer coefficient is discussed.

2 Problem Formulation and Boundary Conditions

The domain for computational study in the present work is shown in Fig. 1. The geometrical dimensions for the domain are considered based on the work of [5]. The secondary fluid is made to enter through the slot of width *D*, into the mainstream with an angle of 35° to the surface. The value of *D* is considered as 5 mm in the present study. The grid required for the computational domain is generated using ANSYS ICEM with non-uniform structured grid, as shown in Fig. 2. Capturing the turbulent

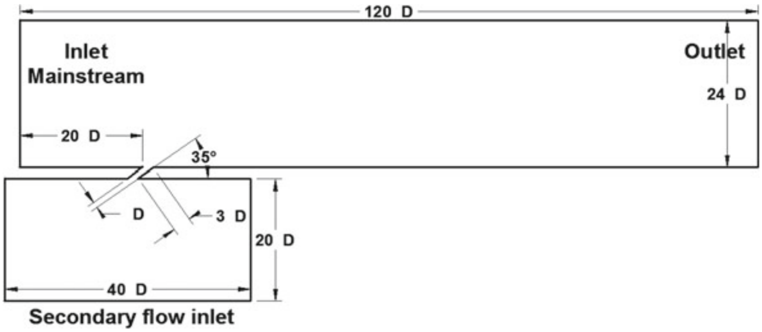


Fig. 1 Geometry of the flow domain considered for the present study

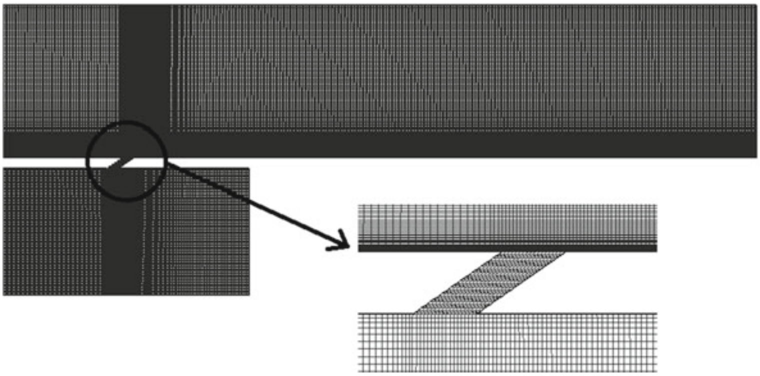


Fig. 2 A structured mesh generated with zoomed view near the mixing region of fluids

boundary layer needs very fine grid size near the wall with a *y*+ value close to unity. The zoomed view in Fig. 2 shows the formation of very fine grid near the wall.

Air is used as working fluid in the present simulation and the solution domain is considered as a 2D, steady, incompressible and turbulent flow. The governing equations solved for continuity, momentum and energy conservation and the Reynolds stress for turbulence are modeled by using Realizable k-ε turbulence model. Second-order upwind interpolation scheme is used for the discretization and equations are solved by using SIMPLE algorithm procedure.

In this study, the velocity and temperature are specified at the inlet of mainstream and outlet is considered as constant zero gauge pressure. The secondary flow is introduced as mass flow inlet into the plenum. A uniform velocity of 20 m/s with a temperature of 300 K is mentioned at the inlet of both primary and the secondary. Turbulence intensity is given 2% with length scale as 1/10th of the inlet extent as mentioned in [5]. Other boundaries are considered as wall and the turbulence scalars are solved by using enhanced wall treatment near wall boundaries.

3 Result and Discussions

3.1 Heat Transfer Coefficient

The heat transfer coefficient is calculated as

$$h = \frac{q''}{(T_w - T_\infty)} \tag{1}$$

where q'' is the heat flux applied on the wall surface and T_w is the computed wall temperature. While calculating the heat transfer coefficient, the temperature of the mainstream and the secondary fluid is maintained equal and is denoted as T_∞. In the case where temperature of primary and secondary flows are different, the fluid temperature has to be replaced by the corresponding reference temperature.

Heat transfer coefficient is represented in terms of ratio of heat transfer coefficient with film cooling to the without film cooling. Figure 3a, b shows the distribution of heat transfer coefficient for flat surface in the downstream direction of injection for blowing ratios of 0.5 and 1.0, respectively. Also, the results are compared with the experimental results of [2] for the case of film cooling through circular holes. When the blowing ratio is very low, the heat transfer is not greatly affected due to the secondary injection. As shown in Fig. 3a, the heat transfer coefficient is nearly equal to with that of without film cooling. The comparison of the present simulation with the experimental result of three-dimensional film cooling shows similar behaviour. In the region immediately downstream of injection, there is a slight decrease in the heat transfer can be observed in both experimental as well as numerical results. The addition of mass flux into the boundary layer results in decreasing the heat transfer near to the injection region, but in the far downstream this effect will disappear making the ratio equal to 1.

In Fig. 3b, a slight decrease in the heat transfer can be observed near the jet exit in the experimental result of [2], but in the present simulation, there is an increase in heat transfer to 1.3 times that of without injection. The reason may be attributed to the spreading of the jet in lateral direction will reduce the velocity of the jet and hence heat transfer would be less.

A comparison of heat transfer coefficient obtained from the present simulation for blowing ratio from 0.5 to 2.0 is shown in Fig. 4. When the blowing ratio is below 1.0, there is a small increase in heat transfer coefficient in the immediate downstream of injection is observed. But at higher blowing ratio, there is significant increase in the heat transfer coefficient is noted compared to the case without injection. Not only near the jet exit but in the far downstream but also heat transfer has increased to almost 1.6–1.8 times higher than that of without injection.

Fig. 3 Heat transfer coefficient distribution along the flat surface form the injection point

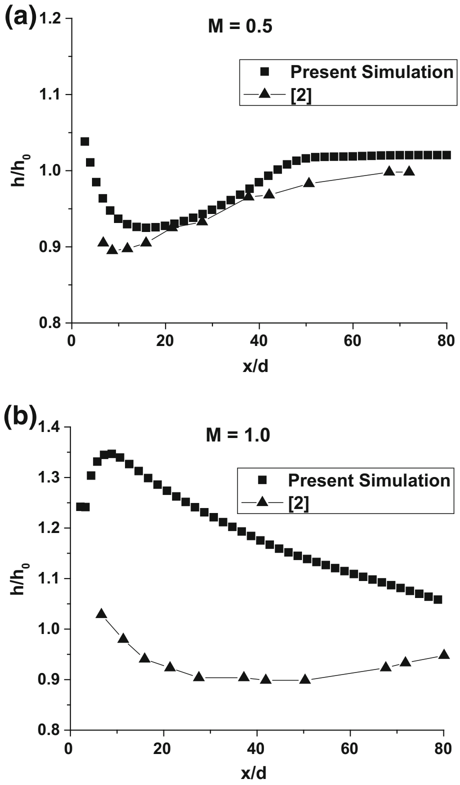
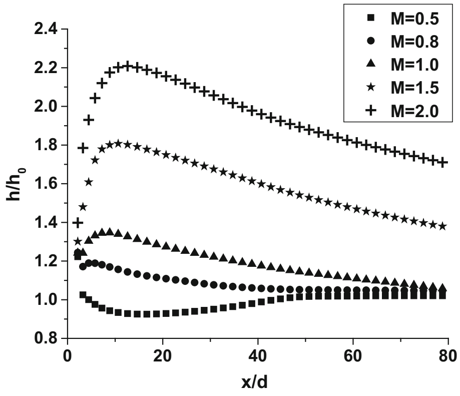


Fig. 4 Comparison of heat transfer coefficient for different blowing ratios



3.2 Turbulence Intensity

One of the major reasons behind the increase in heat transfer coefficient is due to turbulence created at the mixing region. Increase in the blowing ratio will increase

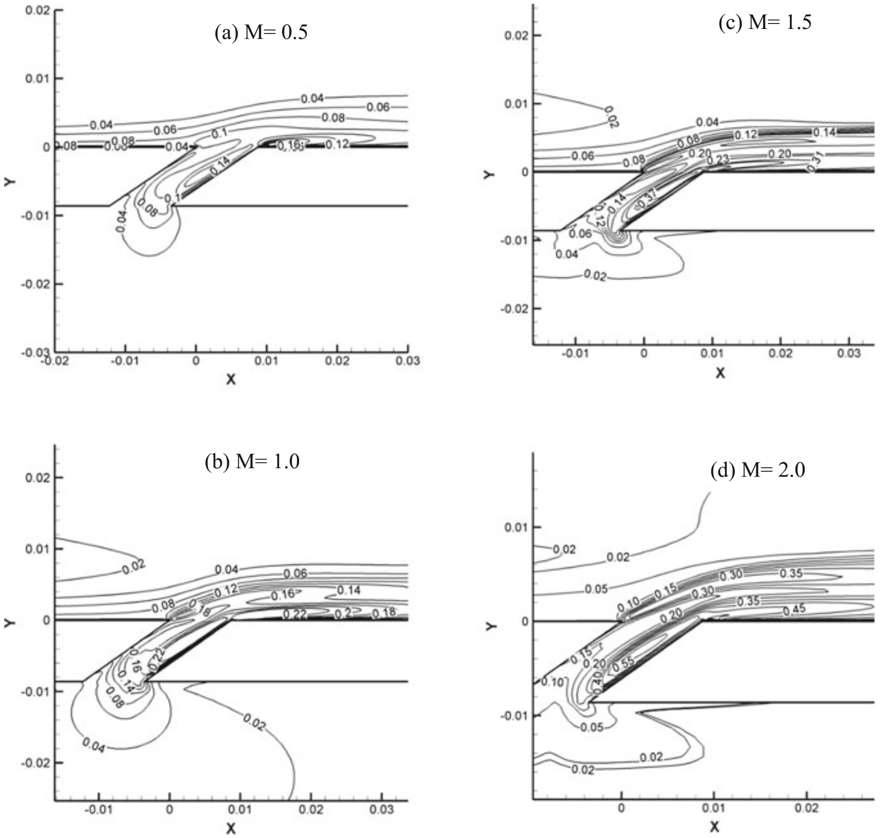


Fig. 5 Turbulence level in the region of interaction of two streams for different blowing ratios

the turbulence due to increased velocity of secondary fluid. And hence heat transfer increases. Figure 5a–d shows the distribution of turbulence intensity in the region of interaction of the two fluid streams for blowing ratios of 0.5, 1.0, 1.5 and 2.0. The turbulence intensity at the free stream is given as 2%. Increased turbulence level is observed at the jet exit and in the immediate downstream region of the flow near the surface.

When the blowing ratio is at 0.5, a slight increase in the turbulence level of 12–14% is observed and it has covered a very small region as shown in Fig. 5a. Since the addition of coolant fluid will reduce the temperature of the boundary layer, at lower blowing ratio, there is a decrease in heat transfer coefficient is identified as shown in Fig. 3a. As the blowing ratio is increased to 1.0, turbulence intensity is also increased to 20–22%. When the blowing ratio is increased to 1.5 and 2.0, there is a drastic increase in the turbulence level is identified and is greater than 30 and 40%, respectively. When Figs. 4 and 5 are compared, it can be clearly observed that as the turbulence intensity is increased, there is an increase in heat transfer coefficient. This increased turbulence level can be attributed to the increase in the blowing ratio.

4 Conclusion

A two-dimensional numerical simulation is carried out for film cooling on flat surface with inclined slot of 35° angle of injection. The film cooling effectiveness and the heat transfer coefficient are investigated for different blowing ratios and results for heat transfer coefficient are compared with the experimental results from the literature. Following conclusions were made from the present study,

- The heat transfer coefficient computed form two-dimensional analysis matches with the experimental results only at lower blowing ratios.
- At higher blowing ratios there is a significant increase in heat transfer coefficient than the experimental results.
- Primary reason behind the increase of heat transfer coefficient is due to increased turbulence intensity at the mixing region of two fluids.
- Increase in the secondary flow velocity induces turbulence in the flow.

References

1. Goldstein, R.J., Eckert, E.R.G., Ramsey, J.W.: Film cooling with injection through holes: adiabatic wall temperatures downstream of a circular hole. J. Eng. Power **90**(4), 384–393 (1968)
2. Eriksen, V.L., Goldstein, R.J.: Heat transfer and film cooling following injection through inclined circular tubes. J. Heat Transf. **96**(2), 239–245 (1974)
3. Kadotani, K., Goldstein, R.J.: Effect of mainstream variables on jets issuing from a row of inclined round holes. J. Eng. Power **101**(2), 298–304 (1979)
4. Bergeles, G., Gosman, A.D., Launder, B.E.: Double-row discrete-hole cooling: an experimental and numerical study. J. Eng. Power **102**(2), 498–503 (1980)
5. Walters, D.K., Leylek, J.H.: A systematic computational methodology applied to a three-dimensional film-cooling flow field. ASME J. Turbomach. **119**(4), 777–785 (1997)
6. Papell, S.S.: Effect on gaseous film cooling of coolant injection through angled slots and normal holes. Technical Note D-299, NASA Lewis Research Center (1960)
7. Fitt, A.D., Ockendon, J.R., Jones, T.V.: Aerodynamics of slot-film cooling: theory and experiment. J. Fluid Mech. **160**, 15–27 (1985)
8. Sarkar, S., Bose, T.K.: Numerical simulation of a 2-D jet-cross flow interaction related to film cooling applications: effects of blowing rate, injection angle and free-stream turbulence. Sadhana **20**(6), 915–935 (1995)
9. Kassimatis, P.G., Bergeles, G.C., Jones, T.V., Chew, J.W.: Numerical investigation of the aerodynamics of the near-slot film cooling. Int. J. Numer. Meth. Fluids **32**(1), 85–104 (2000)
10. Jia, R., Sundén, B., Miron, P., Léger, B.: A numerical and experimental investigation of the slot film-cooling jet with various angles. J. Turbomach. **127**(3), 635–645 (2005)
11. Singh, K., Premachandran, B., Ravi, M.R.: A numerical study on the 2d film cooling of a flat surface. Numer. Heat Transf. Part A Appl. **67**(6), 673–695 (2015)