CFD Modeling of Film Cooling Flow with Inclined Jets

Majid A. Almas[1,2,*]

[1]Department of Mechanical and Materials Engineering, Florida International University, Miami, FL 33199, USA

[2]Department of Marine engineering, King AbdulAziz University, 21589, Saudi Arabia

Abstract: Film cooling has been widely used to control temperature of high temperature and high pressure blades. In a film cooled blade the air taken from last compressor stages is ejected through discrete holes drilled on blade surface to provide a cold layer between hot mainstream and turbine components. A comprehensive understanding of phenomena concerning the complex interaction of hot gasses with coolant flows in a vane passage plays a major role in the definition of a well performing film cooling scheme. In this study turbulent film cooling flow has been studied numerically. The computational simulation is conducted by employing the Reynolds Averaged Navier-Stokes (RANS) approach. The standard $k-\varepsilon$ model with enhanced wall treatment has been implemented for modeling the turbulent flow. Effects of different cooling holes temperature have been studied on the surface of the blade and as results show the temperature of the surface reduces significantly as the temperatures of the cooling holes decreases.

Keywords: Film cooling, inclined jet, RANS model, turbulence model, CFD.

1. INTRODUCTION

Film cooling is an essential technology for the development of high performance gas turbine engines. A well-designed film cooling strategy allows higher turbine inlet temperatures, improving the engine thermodynamic efficiency. A poorly designed strategy can cause high local temperature gradients, leading to component failures and costly repairs. Hence accurate prediction tools are vital for designers. With the increasing complexity of cooling designs, correlations and incremental design approaches have become outdated, signaling the urgent need for “physics-based” tools that can be coupled to standard modern computational tools, such as commercial computational fluid dynamics (CFD) codes. Film cooling is one technique to protect key gas turbine engine components during operation. The fundamental concept behind this strategy can be presented using the following idealized situation: a cool secondary flow is ejected through a series of holes in a given component. Under the appropriate conditions, the flow from the holes coalesces to form a blanket, or film of cooler air that “insulates” the component from an extremely hot mainstream flow. Another expectation is that the addition of “thermal mass” near the wall *via* blowing thickens the boundary layer and acts as a heat sink, consequently reducing heat transfer to the wall [1]. This technique may also be used to protect a surface from large radiative heat loads as well. As an effective cooling technique for gas turbine, film cooling has been widely used to cool the hot components in gas turbine, since it was first introduced into gas turbine cooling system in the early 1950s [3], and film cooling also has become a mainstay of turbine cooling technology nowadays. A detailed review of various shaped holes by Bunker [4] summarized the extent literatures concerning the performance of film cooling holes, and he indicated that film holes geometry was a significant parameter influencing the cooling performance. Gritsch *et al*. [5] compared the performance of the different fan-shaped holes in terms of film cooling effectiveness and heat transfer coefficients. Experiment conducted by Blair [6] aimed to determine the film cooling effectiveness and convective heat transfer coefficient distributions on the endwall of a typical turbine stage. Goldstein *et al*. [7] experimentally investigated the effect of the holes geometry and density of coolant jet on the cooling performance. Graziani *et al*. [8] presented that the distribution of endwall heat transfer coefficients was significantly affected by secondary flows. Pasinato *et al*. [9] experimentally studied the effects of coolant injection from the upstream of the vane stagnation through a series of discrete slots. n. Measurements by Friedrichs *et al*. [10-12] indicated a strong influence of the secondary flow on the film cooling and an effect of the film cooling on the secondary flow, the exit angle of the film holes had little effect on the coolant trajectory. A number of investigations [9-13] indicated that it is a great challenge to cool the regions near the junction of endwall and blade due to the complex vortices. And also recently some work studied the film cooling technique in controlling secondary combustion [14-17]. In this paper turbulent film cooling flow with two inclined

*Address correspondence to this author at theDepartment of Mechanical and Materials Engineering, Florida International University, Miami, FL 33199, USA; Tel: (305) 348-1932; Fax: (305) 348-2569; E-mail: majidaaia@gmail.com

E-ISSN: 2409-5761/16

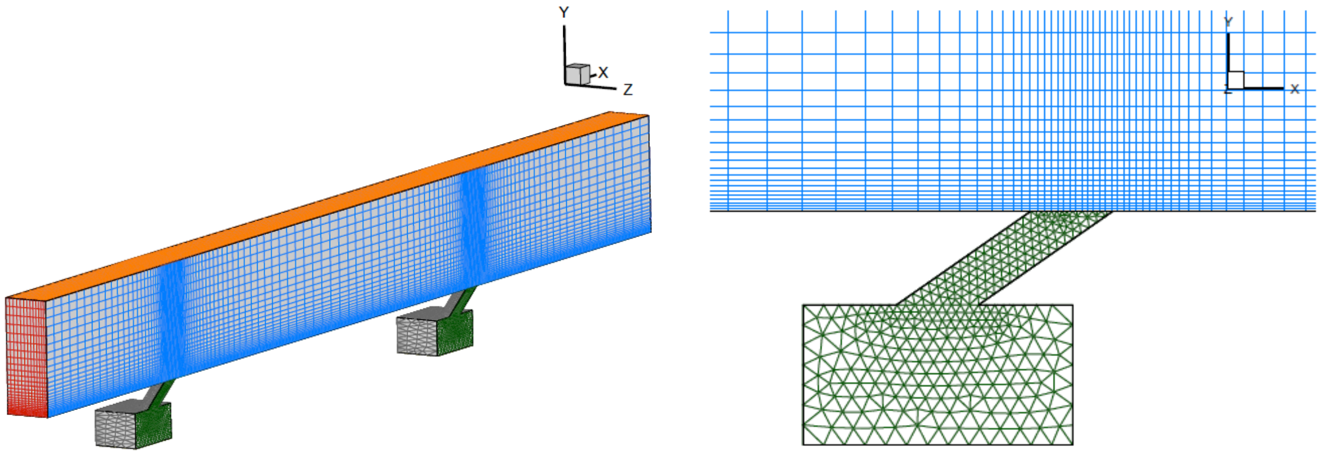

Figure 1a: 3D schematic and grid generation of the problem.

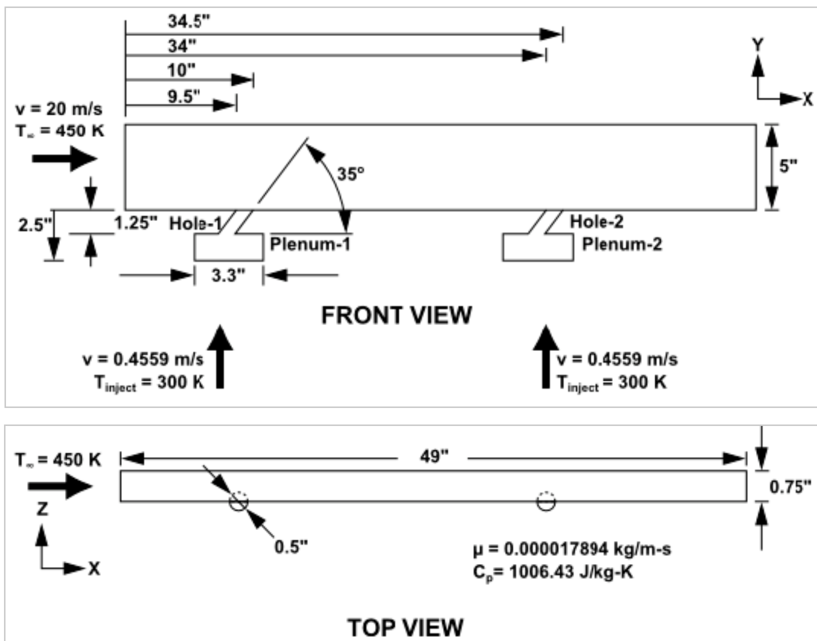

Figure 1b: 2D schematic of the problem.

cooling jets have been studied numerically to investigate the effects of this technique on controlling the extra temperature over the turbine surfaces.

2. PROBLEM DEFINITION

This problem considers a model of a 3D section of a film cooling test rig. The numerical scheme for advection and turbulence kinetic and dissipation is second order upwind. The convergence criteria for continuity, energy and turbulence schemes is 10e-3. A schematic of the problem is shown in Figure **1**. The problem consists of a duct, 49 inches long, with cross-sectional dimensions of 0.75 inches 5 inches. An array of uniformly-spaced holes is located at the bottom of the duct. Each hole has a diameter of 0.5 inches, is inclined at 35 degrees, and is spaced 1.5 inches apart laterally. Cooler injected air enters the system through the plenum having crosssectional dimensions of 3.3 inches 1.25 inches. Only a portion of the domain must be modeled because of the symmetry of the geometry. The bulk temperature of the streamwise air T_∞ is 450K, and the velocity of the air stream is 20m/s. The bottom wall of the duct that intersects the hole array is assumed to be a completely insulated (adiabatic) wall. The secondary (injected) air enters the plenum at a uniform velocity of 0.4559m/s. The temperatures of the injected air (T_{inject}) are 100, 200 and 300K. The properties of air that are used in the model are also mentioned in Figure **1**.

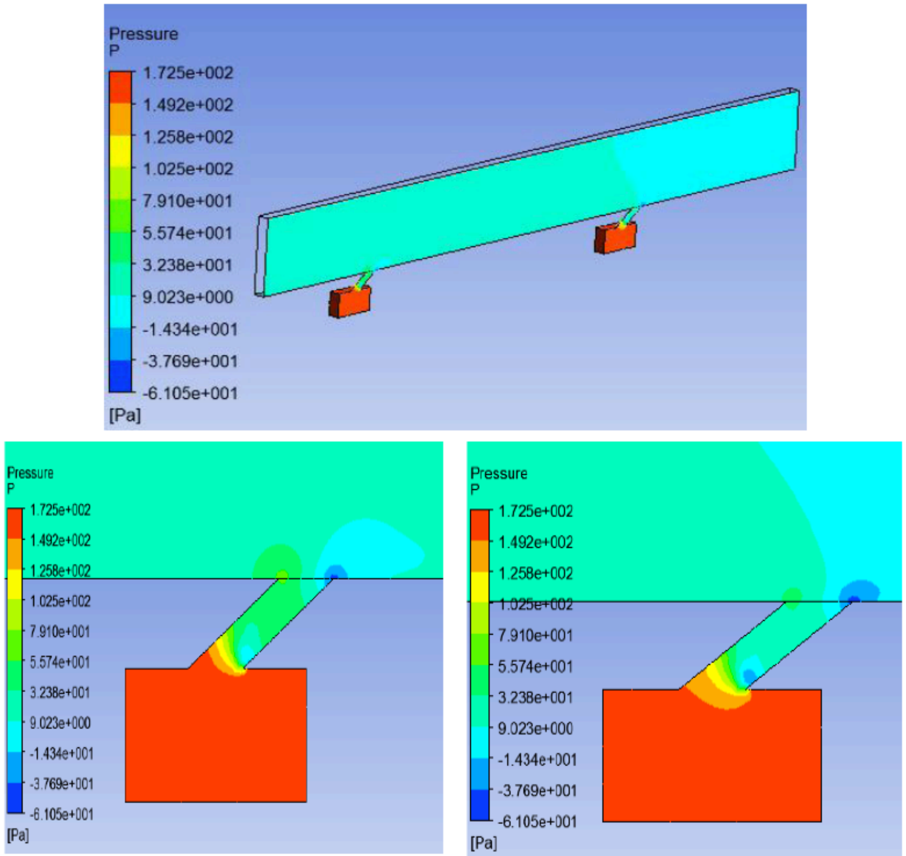

Figure 2: Pressure contour at cooling holes temperatures of 300K.

3. RESULTS AND DISCUSSIONS

Figure **2** shows the variation of the pressure contour for the film cooling flow at temperature of 300K. The pressure is maximum in cooling plenums and has its lowest at the interface surface of cooling hole and the main channel. As can been seen in the figure, the high and low pressure zones are on the upstream and downstream sides of the coolant hole, where the jet first penetrates the primary flow in the duct.

Figure **3** shows the temperature contour for the film cooling flow at temperature of 300K. As can be seen the film cooling technique is significantly crucial in lowering temperature of the blade surfaces. As numerical results show the temperature of the surface reduces from the inlet temperature of main channel which is 450K to almost 300K at the vicinity of the wholes and about 350K at the locations between the two coolant holes. The temperature of the blade surface reduces up to 100K from its initial value at the entrance of the channel which shows the significance of this technique in cooling of the blade surface which has broad applications in such as power plants and aerospace industries where the turbine and compressor

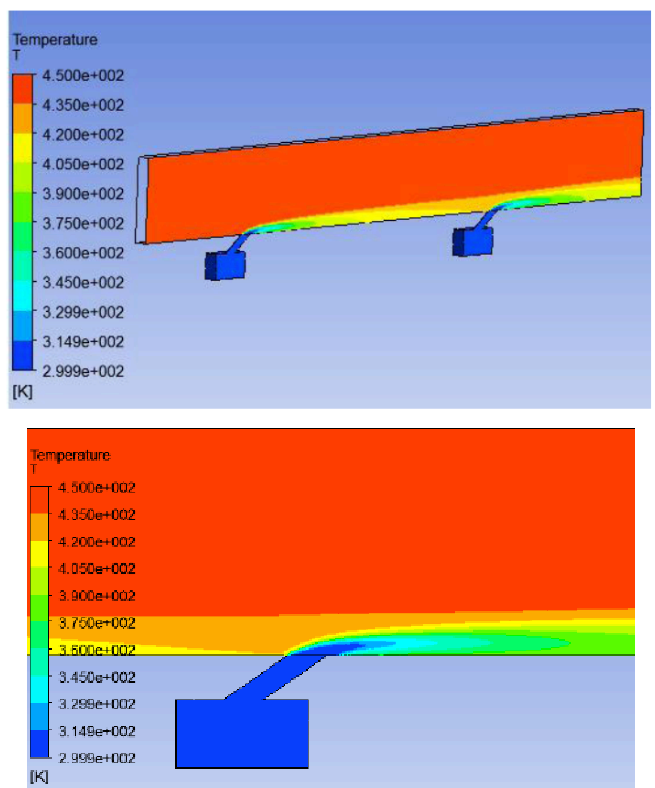

Figure 3: Temperature contour at cooling holes temperatures of 300K.

surfaces of the blades required to be protected from the extra heat. As the results clearly show the coolant flow insulate the bottom of the duct from the higher temperature of the primary flow.

Figure **4** shows variation of the temperature profiles for three different coolant hole temperatures. The figure at the top shows the temperature variation at 100K. As seen in the figure the surface temperature of the bottom of the duct (blade surface) decreases significantly at the vicinity of the coolant hole from its primary value, 450K, to 100K. The temperature is below 350k between the two cooling holes and this value again significantly reduces to 100K at the vicinity of the second coolant hole. And the surface temperature of the blade is around 300K at downstream of the second coolant. The middle figure demonstrate the temperature profile for coolant hole at 200K. Similarly the surface temperature of the bottom of the duct decreases noticeably at the vicinity of the coolant hole from its primary value, 450K, to 200K. The temperature is below 400k between the two cooling holes and this value again significantly reduces to 200K at the vicinity of the second coolant hole. And the surface temperature of the blade is around 350K at downstream of the second coolant. And finally the bottom figure depicts the temperature profile for coolant hole at 300K. Similar to the previous cases the surface temperature of the bottom of the duct decreases sharply at the vicinity of the coolant hole from its primary value, 450K, to 300K. The temperature is below 420k between the two cooling holes and this value again significantly reduces to 400K at the vicinity of the second coolant hole. As seen the surface temperature of the bottom of the duct reduces significantly as the temperature of the coolant holes decreases. At coolant temperature of 100K the bottom surface temperature reduces to 100 and this value reduces to 200 and 300 for coolant temperatures of 200K and 300K respectively. The lower the temperature of the coolant holes the more the blade surface are insulated from the primary temperature value which will result in longer life time of the blades and their results.

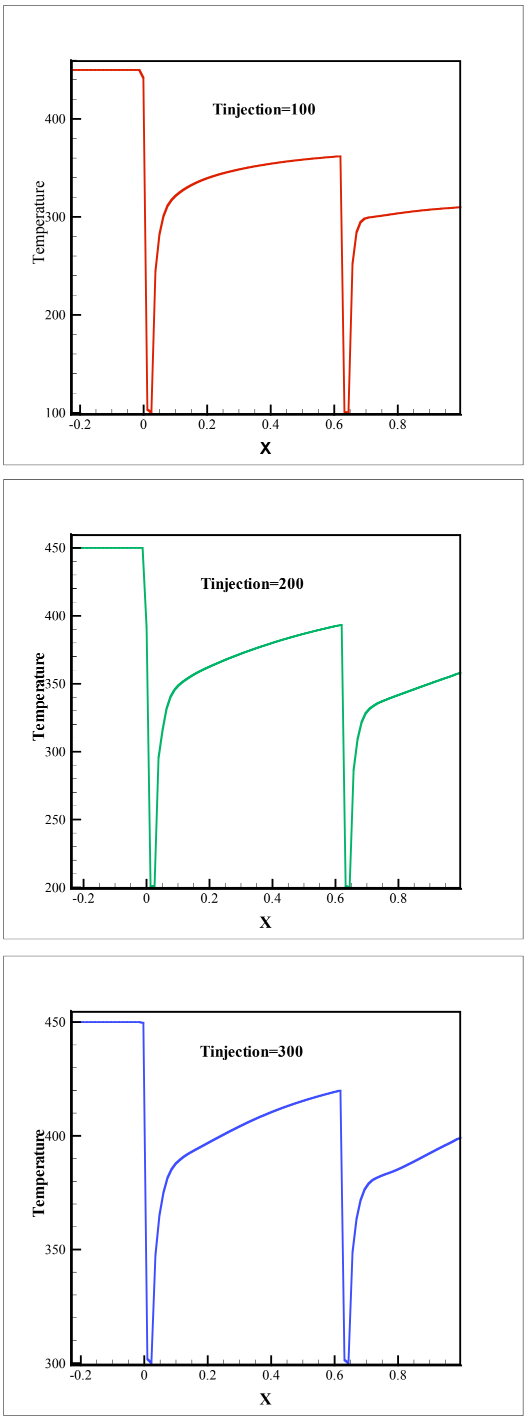

Figure 4: Temperature profiles variations for different cooling hole temperatures of 100, 200 and 300K.

5. CONCLUSIONS

In this study a numerical modeling of a film cooling technique has been investigated. The $k-\varepsilon$ turbulence model [18] has been implemented for simulation of such complex turbulent flow. The numerical results have been shown and described through graphs. The effects of different coolant hole temperatures have been studied on the surface temperature of the bottom of the duct. Results revealed that the film cooling technique is significantly important in reducing the temperature of the blade surfaces. And also decreasing the coolant hole temperatures decreases the primary value of the surface temperature significantly which results in better protection of the blade surfaces.

ACKNOWLEDGMENTS

The author would like to thank both the Saudi Arabian Cultural Mission in Washington D.C. and King Abdulaziz University in Jeddah, KSA for their support.

REFERENCES

[1] Moffat R. Turbine Blade Cooling. Heat Transfer and Fluid Flow in Rotating Machinery 1987; 3-36.

[2] Bunker RS. Film cooling: breaking the limits of diffusion shaped holes, Heat Transfer Res 2010; 41: 627-650. http://dx.doi.org/10.1615/HeatTransRes.v41.i6.40

[3] Bunker RS. A review of shaped hole turbine film-cooling technology, ASME J Heat Transfer 2005; 127: 441-453. http://dx.doi.org/10.1115/1.1860562

[4] Gritsch M, Schulz A, Wittig S. Adiabatic wall effectiveness measurements of film-cooling holes with expanded exits. ASME Paper 1997; 97-GT-164.

[5] Blair MF. An experimental study of heat transfer and film cooling on a largescale turbine endwalls. ASME J Heat Transfer 1974; 96: 524-529. http://dx.doi.org/10.1115/1.3450239

[6] Goldstein R, Eckert E and Burggraf F. Effects of hole geometry and density on three-dimensional film cooling. Int J Heat Mass Transfer 1974; 17: 595-607. http://dx.doi.org/10.1016/0017-9310(74)90007-6

[7] Graziani RA, *et al*. An experimental study of endwall and airfoil surface heat transfer in a large scale turbine blade cascade. ASME J Gas Turbines Eng Power 1980; 102: 257-267. http://dx.doi.org/10.1115/1.3230246

[8] Pasinato HD, Squires KD and Roy RP. Measurement and modeling of the flow and heat transfer in a contoured vane-endwall passage. Int J Heat Mass Transfer 2004; 47: 5685-5702. http://dx.doi.org/10.1016/j.ijheatmasstransfer.2004.07.032

[9] Friedrichs S, Hodson HP and Dawes WN. Distribution of film-cooling effectiveness on a turbine endwall measured using the ammonia and diazo technique. ASME J Turbomach 1996; 118: 613-621. http://dx.doi.org/10.1115/1.2840916

[10] Friedrichs S, Hodson HP and Dawes WN. Aerodynamic aspects of endwall filmcooling. ASME J. Turbomach 1997; 119: 786-793. http://dx.doi.org/10.1115/1.2841189

[11] Friedrichs S, Hodson HP and Dawes WN. The design of an improved endwall film-cooling configuration. ASME J Turbomach 1999; 121: 772-780. http://dx.doi.org/10.1115/1.2836731

[12] Yamao H, Aoki K, Takeishi K and Takeda K. An experimental study for endwall cooling design of turbine vanes. IGTC-1987 Tokyo, Japan 1987.

[13] Knost DG and Hole KA. Adiabatic effectiveness measurements of endwall filmcooling for a first stage vane. ASME J Turbomach 2005; 127: 297-305. http://dx.doi.org/10.1115/1.1811099

[14] Ghasemi E, Soleimani S and Lin CX. Control of turbulent combustion flow inside a gas turbine combustion chamber using plasma actuators. Fuels, Combustion and Material Handling, ASME Power and Energy, (doi:10.1115/POWER2015-49499).

[15] Ghasemi E, Soleimani S and Lin CX. Secondary reactions of turbulent reacting flows over a film-cooled surface. Int Commun heat Mass transfer 2014; 55: 93-101. http://dx.doi.org/10.1016/j.icheatmasstransfer.2014.04.007

[16] Ghasemi E, Soleimani S and Lin CX. RANS simulation of methane-air burner using local extinction approach within eddy dissipation concept by OpenFOAM. Int Commun heat Mass transfer 2014; 54: 96-102. http://dx.doi.org/10.1016/j.icheatmasstransfer.2014.03.006

[17] Ghasemi E, Soleimani S, Almas MA. Finite Element Simulation of Jet Combustor Using Local Extinction Approach within Eddy Dissipation Concep. J Adv Therm Sci Research 2014; 1: 57-65. http://dx.doi.org/10.15377/2409-5826.2014.01.02.4

[18] Ghasemi E, McEligot DM, Nolan KP and Crepeau J. A Tokuhiro, RS Budwig, Entropy generation in a transitional boundary layer region under the influence of freestream turbulence using transitional RANS models and DNS. Int Commun heat Mass transfer 2013; 41: 10-16. http://dx.doi.org/10.1016/j.icheatmasstransfer.2012.11.005

Received on 24-05-2016 Accepted on 26-05-2016 Published on 14-07-2016

DOI: http://dx.doi.org/10.15377/2409-5761.2016.03.01.5