International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:4, No:2, 2010

Cooling Turbine Blades using Exciting Boundary Layer

Ali Ghobadi, Seyed Mohammad Javadi, and Behnam Rahimi

***Abstract*—**The present study is concerned with the effect of exciting boundary layer on cooling process in a gas-turbine blades. The cooling process is numerically investigated. Observations show cooling the first row of moving or stable blades leads to increase their life-time. Results show that minimum temperature in cooling line with exciting boundary layer is lower than without exciting. Using block in cooling line of turbines' blade causes flow pattern and stability in boundary layer changed that causes increase in heat transfer coefficient. Results show at the location of block, temperature of turbines' blade is significantly decreased. The k-ε turbulence model is used.

***Keywords*—**Cooling, Exciting Boundary Layer, Heat Transfer, Turbine Blade.

I. Introduction

THE ability of today's gas turbine engines to withstand increasingly higher turbine-inlet temperatures has been largely due to the advancement in cooling technology. Increasing or decreasing heat transfer is concerned in the world. Growing demand to extract heat from limited areas, conducts engineers to develop techniques to enhance heat transfer coefficient. One of old methods is to augment turbulent intensity in the boundary layer by roughening the surface and motivating turbulent heat transfer [1].

Other method suggests wavy surfaces by which again many small vortices generate and flow into the boundary layer, helping to mix particles and therefore to enhance heat transfer [2]. These methods are widely used in heat exchangers and various shapes and designs are presented by Keys and London [3]. In all of these cases the amount of augmentation of heat is attractive while the increase in friction coefficient leaves a limited era to employ the technique and is preventive.

Suzuki [4], employed an insert into turbulent boundary layer and measured both heat and skin friction coefficient along the affected area. Amazingly results were violating Reynolds analogy between heat and momentum transfer, i.e. while the insert was a cause of heat transfer enhancement (HTE), the skin friction was reduced simultaneously. More study by Teraguchi [5] and Suzuki [6] are also reported the same type of violation in Reynolds analogy. However, if the result is proved to be sustainable in similar stimulation of boundary layer, this would be unique advantage of HTE by an insert. Present work insists on maximizing heat transfer coefficient, while skin friction is constantly minimized and blockage effect by obstacle is also diluted. Therefore the problem is one of three objectives optimization. Schmidt et al. [6] examined film cooling performance of 60_ compound angled holes on a flat plate surface, with and without forward expanded shaped exit, and compared that with axial cylindrical holes. Dittmar et al. [7] conducted measurements on suction side of turbine guide vanes inside a wind tunnel. To study the adiabatic film cooling effectiveness and heat transfer coefficient four different cooling hole configurations – double rows of cylindrical holes, double rows of discrete slots, a single row of axial fan-shaped holes, and a single row of compound angle fan-shaped holes were chosen. Mhetras et al. [8] observed the excellent coolant coverage offered by compound shaped holes near the tip region of the pressure side. For increasing efficiency, temperature increases, so proper materials are needed for creating blades and appropriate cooling is most helpful. In this paper, heat transfer at inlet of turbines' blades and the effect of excited boundary layer on cooling turbines' blade is investigated.

II. Governing Equation

The average gas phase equations are as follows:

-Continuity:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

-Momentum:

$$\left[\frac{\partial}{\partial x}(\rho uu)+\frac{\partial}{\partial y}(\rho uv)\right]=-\frac{\partial p}{\partial x}+\mu\nabla^2 u -\rho\overline{u'u'}-\frac{\partial}{\partial y}(\rho\overline{u'v'})-\frac{\partial}{\partial z}(\rho\overline{u'w'}) \tag{2}$$

$$\left[\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho vv)\right]=-\frac{\partial p}{\partial y}+\mu\nabla^2 v -\rho\overline{v'v'}-\frac{\partial}{\partial x}(\rho\overline{u'v'})--\frac{\partial}{\partial z}\rho\overline{v'w'} \tag{3}$$

A. Ghobadi is with Mechanical Engineering Department, Sarkhoon and Ghashm gas treating Co, Iran (e-mail: ghbd_2@yahoo.com).

S. M. Javadi and B. Rahimi are with Mechanical Engineering Department, Ferdowsi University of Mashhad, Iran (e-mail: mohammad.javadi@gmail.com, behnam112@yahoo.com).

$$\left[\frac{\partial}{\partial x}(\rho uw)+\frac{\partial}{\partial z}(\rho vw)\right]=-\frac{\partial p}{\partial z}+\mu\nabla^2 w - \frac{\partial}{\partial x}(\rho\overline{u'w'})-\frac{\partial}{\partial y}(\rho\overline{v'w'})-\rho\overline{w'w'} \tag{4}$$

The simplest "complete models" of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard $k-\varepsilon$ model falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [9]. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism. The standard $k-\varepsilon$ model [10] is a semi-empirical model based on model transport equations for the turbulence kinetic energy k and its dissipation rate (ε). The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In the derivation of the $k-\varepsilon$ model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard $k-\varepsilon$ model is therefore valid only for fully turbulent flows. The turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations[10]:

$$\frac{\partial(\rho k)}{\partial t}+div\,(\rho kU)=div\left[\mu+\frac{\mu_t}{\sigma_k}gradk\right]+ G_k+G_b-\rho\varepsilon-Y_M+S_K \tag{5}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+div\,(\rho\varepsilon U)=div\left[(\mu+\frac{\mu_t}{\sigma_\varepsilon})grad\,\varepsilon\right] +C_{1\varepsilon}\frac{\varepsilon}{K}(G_K+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \tag{6}$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}, C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. S_k and S_ε are user-defined source terms. The turbulent (or eddy) viscosity, μ_t, is computed by combining k and ε as follows[10]:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{7}$$

Where C_μ is constant.

The model constants $C_{1\varepsilon}, C_{2\varepsilon}, C_\mu, \sigma_k$ and σ_ε have the following default values [9]:

$$\sigma_{1\varepsilon}=1.44\ \sigma_{2\varepsilon}=1.92\ \sigma_k=1\ C_\mu=0.09\ \sigma_\varepsilon=1.3$$

III. Flow Simulation

The geometry of the air condenser, that is generated using Solid works™ software, and its boundary conditions are displayed in Fig. 1. Different grid sizes have been tested, Grid independency has been verified (Fig. 2) and finally a structured grid, that is generated using Gambit™ 2.3.16 software, offers the best compromise between precision and computational effort. For solving heat transfer equation, the blade is divided into 34,332 quadrilateral cells and the cooling way is divided into 3,207 quadrilateral cells. When out flow converged, for increasing efficiency, cells of cooling way are doubled. Simulations are performed with the commercial CFD software Fluent™ [11]. The resultant systems of discretised linear algebraic equations are solved by using the density-based explicit solver [11]. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) [12] algorithm is used for the pressure-velocity coupling, whereas the power-law [12] scheme is used for the convection-diffusion formulations. Fluid flow rate is 1.5kg/s and flow rate at the inlet of cooling way is 0.00097kg/s. In Fig. 3, the location of block in cooling line of turbine's blade is shown. The 2mm diameter cylindrical block is placed in a 5mm diameter pipe.

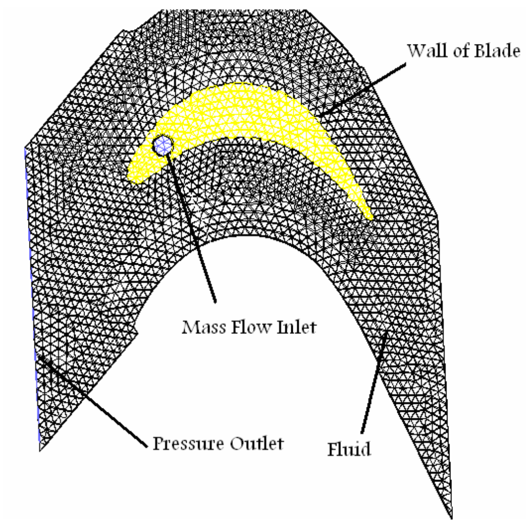

Fig. 1 Geometry and the boundary conditions

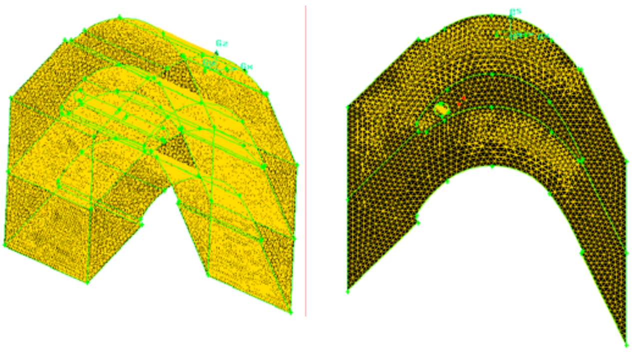

Fig. 2 Structured grid generated

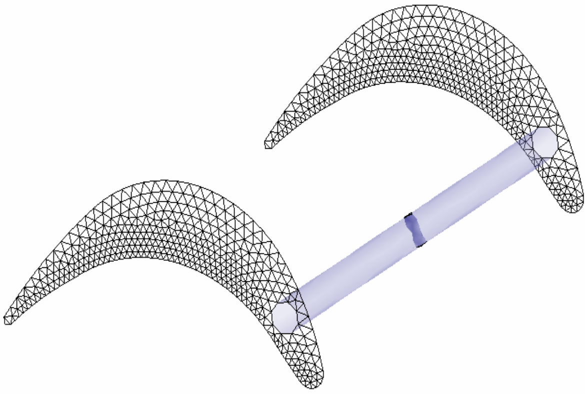

Fig. 3 The location of the block in cooling way

IV. Results

In this paper, heat transfer rate in cooling line of turbine's blade with and without exciting boundary layer is investigated. For more accuracy, the fluid near the blade is simulated. Temperature distribution is showed in Figs. 4 and 5 with inlet flow rate equal to 0.4kg/s, with and without exciting boundary layer. As expected, in without exciting case, temperature of blade's surface in inlet area of the cooling way has its minimum value. As moving toward outlet area, temperature of blade's surface is increasing due to increase in cooling fluid temperature. But, heat transfer rate increase due to the turbulence effects in the exciting case. As a consequence, minimum temperature occurs in middle way near block area instead of inlet area.

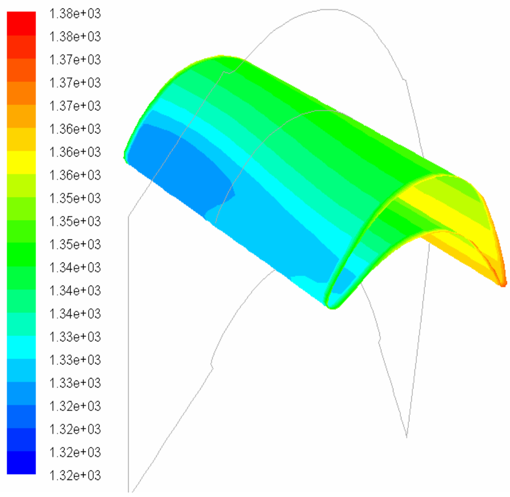

Fig.4 temperature distribution(k) on the surface of blade without excitation

Fig. 6 and Fig. 7 depict the variation of temperature in cooling way with and without exciting boundary layer. It is observed that there is much less temperature increase in exciting case rather than non-exciting case. It can be concluded, without exciting boundary layer, minimum temperature increases during cooling way. But with exciting boundary layer, minimum temperature, which is 18 centigrade lower than without excitation, is occurred near the block area.

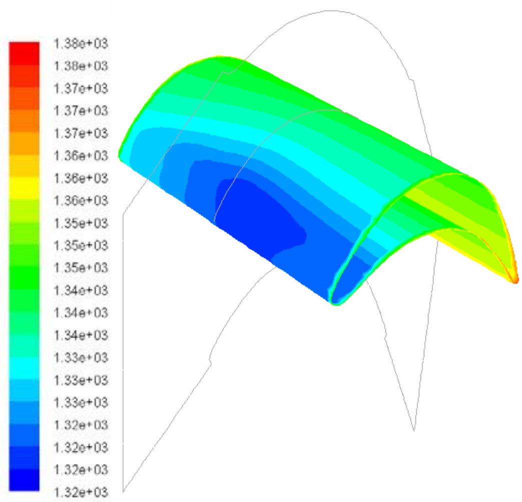

Fig. 5 Temperature distribution(k) on the surface of blade with excitation

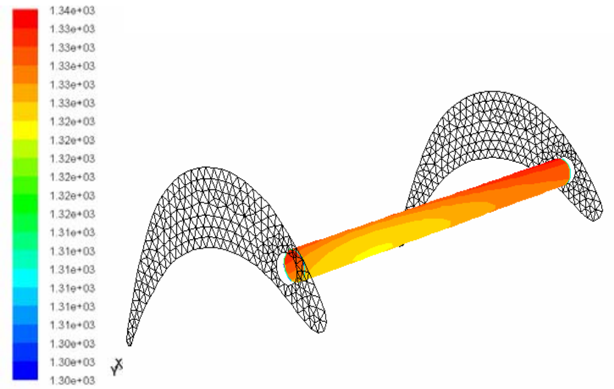

Fig. 6 Temperature distribution(k) in the cooling way without excitation

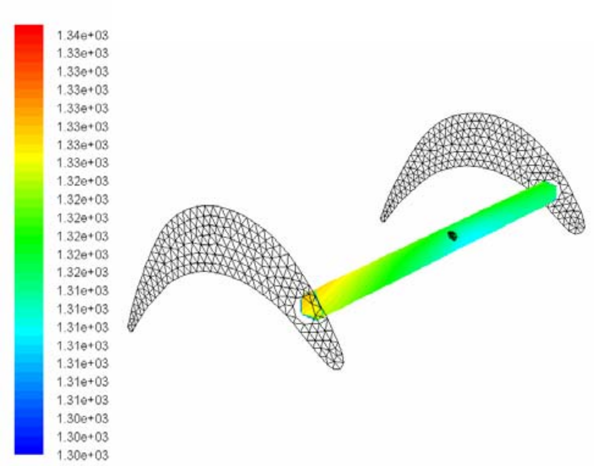

Fig. 7 Temperature distribution(k) in the cooling way without excitation

Proper arrangement leads to increase the efficiency of cooling. Our investigation shows that third cylindrical block arrangement has lower pressure drop and high excitation intensity. Fig. 11 shows the different location of blades with third cylindrical block arrangement. Analyzing heat transfer rate from blade to cooling fluid for two cases, with and without exciting, shows ten percent enhancement in heat transfer rate. In exciting case, heat transferred from blade is 222 watt and in non-exciting case, it is 202 watt.

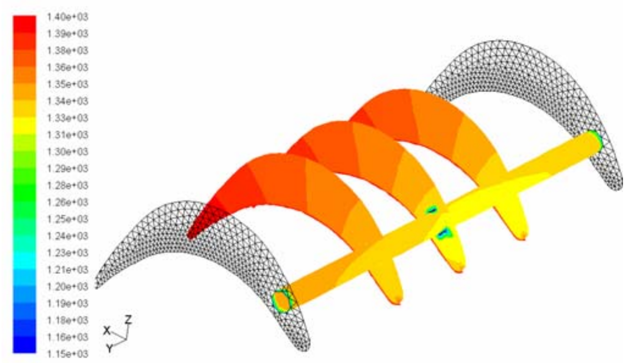

Fig. 12 Temperature distribution (k) in different location of blades with exciting boundary

V. Conclusion

In this paper the effect of exciting boundary layer on heat transfer in turbine blade have been investigated. Results show:

- Using block in cooling line in turbines' blade, heat transfer coefficient increases.
- Minimum of temperature is occurred near the block.
- Minimum temperature in cooling line with exciting boundary layer is lower than without using it.
- With using appropriate arrangement, heat transfer is increased without high decreasing in pressure.

References

[1] Perry L. Young, Surface roughness effects on heat transfer in micro scale single phase, Proceedings of the Sixth International ASME Conference on Nano channels, Micro channels and Mini channels, ICNMM2008, June 23-25, 2008, Darmstadt, Germany
[2] Y. Suzue, K. Morimoto, N. Shikazono, Y. Suzuki and N. Kasagi, High performance heat exchanger with oblique wave walls, 13th International Heat Transfer Conference, Sydney, Australia, 13-18 August 2006..
[3] A. M. Jacobi, Y. Park, Y. Zhong, G. Michna, and Y. Xia, High performance heat exchangers for air-conditioning and refrigeration applications (non-circular tubes), Report No. ARTI-21CR/605-20021-01, July 2005, University of Illinois.
[4] K. Inakova, J.Yamamoto, K.Suzuki, Dissimilarity Between Heat Transfer and Momentum Transfer in a Disturbed Boundary Layer with Insertion of a Rod - Modeling and Numerical Simulation, International Journal of Heat and Fluid Flow, 20(1999) 290-301.
[5] K.Teraguchi, K.Katoh, T. Azuma, Dissimilarity between Turbulent Momentum and Heat Transfer by Excitation of Transverse Vortex, Journal of the Japan Society of Mechanical Engineers, 2005 (80).
[6] D.L. Schmidt, B. Sen, D.G. Bogard, Film cooling with compound angle holes: adiabatic effectiveness, ASME Paper No. 94-GT-312, 1994.
[7] J. Dittmar, A. Schulz, S. Wittig, Assessment of various film cooling configurations including shaped and compound angle holes based on large scale experiments, ASME Paper No. GT-2002-30176, 2002.
[8] S. Mhetras, D. Narzary, Z. Gao, J.C. Han, Effect of a cutback squealer and cavity depth on film-cooling effectiveness on a gas turbine blade tip, AIAA Paper No. AIAA 2006-3404, 2006.
[9] B. E. Launder, D. B. Spalding. Lectures in Mathematical Models of Turbulence. Academic Press, London, England, 1972.
[10] T.B Gatski, M.Y.Hussaini, J.L. Lumley, Simulation and Modeling of Turbulent flows, Oxford University Press, New York, 1996.
[11] Fluent User's Guide, Version 4.3, vols, 1-4, Fluent Incorporated, Lebanon, NH, 1995.
[12] S. V. Patankar, Numerical Heat Transfer and Fluid Flow, McGraw-Hill, 1980.