ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

# Numerical Investigation of Film Cooling For a Section of High Pressure Turbine Blade

**Kandi Hemalatha**
Center for Nano Technology,
Andhra University,
Visakhapatnam, India.

**M. Prudvi Raj Akshay Sai Kumar**
Department of Mechanical Engineering,
Indian Institute of Technology,
Kharagpur, India.

**Prof. Ramji**
Department of Mechanical Engineering,
Andhra University College of Engineering (A),
Visakhapatnam, India.

**Trushar B. Gohil**
Department of Mechanical Engineering,
Visvesvaraya National Institute of Technology,
Nagpur, India.

## ABSTRACT

***For higher efficiencies of a gas turbine, at the inlet of turbine it should function at higher temperatures. Development of higher temperature is limited by materialistic properties. Hence, methods must be devised for tackling this problem; one such method is film cooling. In this method, cold air from compressor is ducted through internal chambers of turbine blade and discharged on to the surface to form an insulating blanket over the turbine blade. Turbine blade (NACA-s1091) coordinates are taken from the airfoil tool data base and are validated with xflr5 v6.10.03 data using ANSYS FLUEN. A new parameter called step (S) = 12.5% with maximum thickness of blade is used for film cooling, for Reynolds number (Re) = 1, 00,000 and angle of attack (AoA) =5.25$^0$ for Blow ratios (M) =0, 0.5, 1.5 and 1.633. It is observed that as the Value of M is increased, the effectiveness of film cooling of the defined geometry is also increased which will facilitate the higher working temperature of the blade. Aerodynamic drag is also expected to reduce due to the setup.***

*Key words:* ***ANSYS (FLUENT), step, Blow ratio, Film cooling, Film cooling effectiveness;***

## INTRODUCTION

Gas turbines are operated continuously under stress for a long operational time period. Demand for increasingfor the performance of turbine is met by increasing the temperature which leads to decrease in endurance limit of the material of turbine blade, which decreases the life time of the turbine blade. Hence, there is a need to introduce the cooling mechanism to protect the blade.

Development and testing of a new film cooling hole geometry, the converging slot-hole or console. Both the thermal and aerodynamic performance was measured, using the adiabatic effectiveness and heat transfer coefficient and aerodynamic loss respectively, to quantify performance[2]. In a similar study [3], the film hole is angled at 30$^0$ to the cross flow with a Reynolds number of 17,400. Investigation was about advanced cooling hole geometries on film cooling effectiveness over flat surface.Experimentsconducted on film cooling effectiveness of a 3-D gas turbine end wall with one fan-shaped cooling hole[4]. The simulations were performed for adiabatic and conjugate heat transfer models.

Turbulence closure was investigated using three different turbulent models namely realizable k-ε model, SST k-ω and v2-f turbulence model. It was reported that the realizable k-ε turbulence model exhibited the best agreement. In another study [5], it is observed that heat transfer coefficient vary with the hole orientation angle from stagnation line and the film cooling effectiveness increases with the increase in blowing ratio.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

## Methodology

Computational Fluid Dynamics is part of fluid mechanics which uses numerical method to solve and analyze problem related to fluid flow. CFD approach is one of the effective method for solving the fundamental non-linear equations that rule the fluid flow, heat transfer and turbulence of flow.

In the present work numerical simulation is carried out by using ANSYS FLUENT software and the governing equation which are used in this CFD analysis are conservation of mass, momentum and energy. For unsteady flow, the fluid properties are function of time.Density based, coupled implicit algorithm is used for the solution of governing equations. As the flow was turbulent, the numerical analysis was done by using density based k-ω SST turbulence model.

The working fluid in this domain is air which is assumed to obey the ideal gas law and the viscosity of air was taken from Sutherland curve. Advection terms in all equations is discretized by second ordered upwind scheme. The variables used in this Analysis named Blowratio (M), Effectiveness (η) is defined as follows.

$$\text{Blow ratio (M)} = \boldsymbol{\rho}_c v_c / \boldsymbol{\rho}_i v_i$$

Where $\boldsymbol{\rho}_i$,$\boldsymbol{\rho}_c$,$v_i$, and $v_c$are densities and velocities of air from inlet and step respectively.

$$\text{Effectiveness, } (\eta) = (T_i - T)/(T_i - T_c)$$

Where, $T_i$, $T_c$, T are temperatures of air at inlet, step and blade surface respectively.

## Geometry and Meshing

Figure 1 shows the schematics drawing of the present computational domain. For the development of blade shape the available NACAs-1091 blade's data co-ordinates are imported and oriented at AoA=5.25$^0$ in ICEM. The step is created with 12.5% of maximum thickness located at 13.4% of cord length. For the mesh generation ICEM-CFD software is used and structured mesh of elements of 65,225 is developed with near wall mesh refinement of y+ =1.0 (Figure 1).

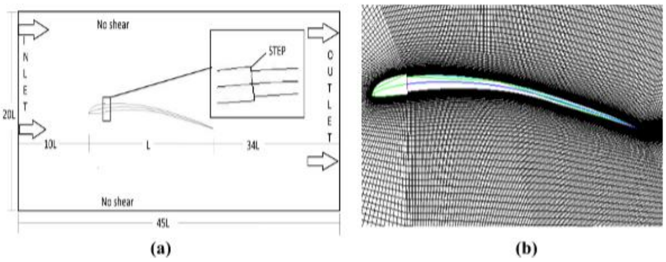


**Figure 1: (a) Schematic drawing of present computational domain with NACA (S1091) airfoil and (b) meshing configuration near the airfoil geometry.**

## Boundary conditions

In the present work, fluid passing through the domain was considered as an ideal gas with uniform velocity of 15.729 m/s (Re = 1,00,000) at temperature of 1200 K. Turbulence intensity was specified at the inlet of 1% and hydraulic diameter of 1 m at inlets (Inlet of the domain and at step inlet). At the blade surface no-slip boundary condition were considered with adiabatic wall condition. Cross streamwise domain boundaries are considered as free slip and with adiabatic wall. Domain outlet is taken as pressure outlet at atmospheric pressure as shown in Table1. The M is set to 0, 0.5, 1.5 and 1.633 for the step geometry and the corresponding step inlet velocities are 0, 2.993, 5.987, 8.981 and 9.777 m/s. The temperature is set to 450 K for the cooled air from step inlet.

**Table1: Boundary conditions**

| Parameters | Input values |
|---|---|
| Analysis type | Density based, steady state, implicit method |
| Turbulence model | k-ω (SST) |
| Material | Air ideal Gas |
| Inlet | Velocity inlet |
| Blade surface | No slip, an adiabatic |
| Domain boundaries | Free slip, an adiabatic |
| Step inlet | Velocity inlet |
| Outlet | Pressure outlet |

## Problem definition

To alter the geometry by using newly introduced entity called step (Figure 1(a)) and to blow cooled compressor air through it at different blow ratios (M). An analysis has been carried out on NACA-s1091 blade at Re =

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

1,00,000 and AoA = $5.25^0$ and verify an aerodynamic and thermal performance of the step on the down-stream of the airfoil.

## Results and Discussion

### Validation:

For validation of present numerical method, airfoil geometry without step was considered. Coefficients of lift ($C_L$) and drag ($C_D$) with Xflr5 v6.10.03 data based was compared and reported in Table 2 this simulation shows good agreement with the available data.

### Table 2: validation results

| Parameters | Validated | computational | % Error |
|---|---|---|---|
| $C_D$ | 1.662e-02 | 1.876e-02 | 11.4 |
| $C_L$ | 1.123e00 | 1.089e00 | 3 |

Based on these simulations the performanceof above mentioned boundary conditions, an increase in Blow ratio (M) had following effects.

### Temperature on blade surface:

Figure 2 visualizes the temperature distribution in vicinity of the blade. This shows us that a single step can provide cooling more effectively throughout the downstream till the trailing edge compared to small regions covered by film cooling using holes in studies reported in the literatures.

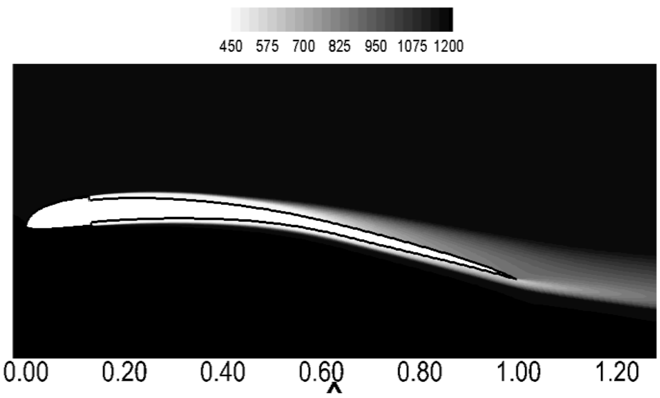


Figure 2: Temperature distribution for blow ratio M=1.5

### Effectiveness:

On suction side and pressure side of the blade are respectively shown in Figure 3. It is observed as the M increased theeffectiveness is increased which reflect the effect of film cooling near the airfoil wall. Figure 3 also shows that because of high temperature at tip end of the airfoil pick is observed at x = 1.0. Here, maximum blow ratio M=1.633 shows the maximum effectiveness on both side of the blade.

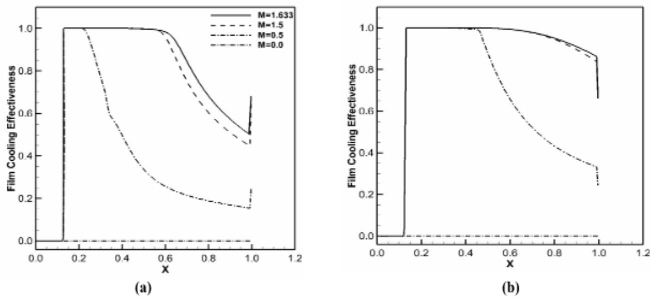


Figure 3: Variation of effectiveness over a length of the airfoil on (a) suction side and (b) pressure side.

### Flow separation:

Flow separation occurs when the boundary layer travels moves away against an adverse pressure gradient with the speed of the boundary layer relative with respects to the object falls almost to zero. The fluid flow which has detached from the surface of the object will form eddies and vortices. Figure 4 shows the flow separation and formation of separated vortex near the step for M = 0.0. It is observed that the separated vortex move down stream with increased in the M.

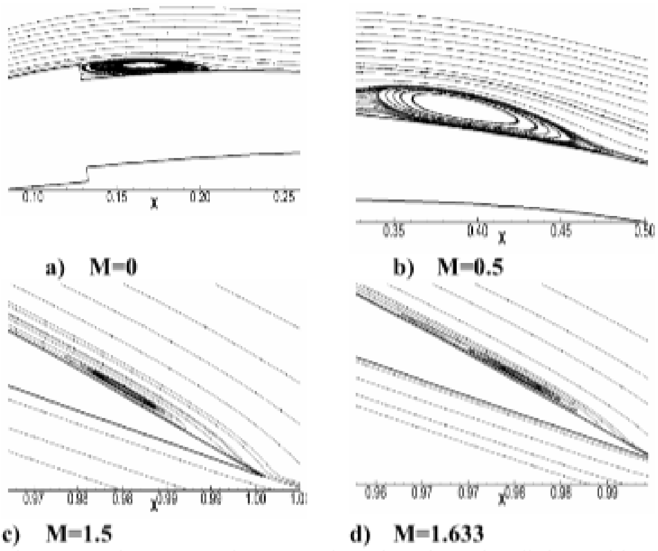


Figure 4: Flow separation at various locations for different blow ratios

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

### Coefficient of drag (CD):

Table3 depicts different values of drag attained for different blow ratios (M). For reference $C_D$ value of the blade NACA s1091 is 1.662e-02. This decrease in drag facilitates us for using this setup for angle of attacks where $C_L$ and $C_D$ are high but their ratio is low due to higher $C_D$.

**Table 3: $C_D$ obtained for different Blow ratios (M)**

| Blow ratio(M) | $C_D$ obtained |
|---|---|
| 0 | 1.715e-02 |
| 0.5 | 1.803e-02 |
| 1.5 | -3.123e-02 |
| 1.633 | -4.181e-02 |

### Conclusion

Numerical investigations have been applied on film cooling for the step region. Using compressible Navier-stokes equation, k-ω SST turbulence models. The considered Reynolds number is of 1,00,000 and M varies from 0 to 1.633 for better film cooling effect. The analysis derives the following conclusions:

- Increasing the Blow ratio (M) increases the effectiveness of cooling significantly up to the tailing edge, also the vertices location shift down-stream.
- Thrust developed due to ejection of air through step has noticeable effect against drag.

### Future scope

This methodology is applicable for 3D turbine blade section also and further, reducing the temperature Nano level thermal barrier coating will be applied.

### Acknowledgements

Authors are grateful to the Department of Mechanical Engineering, Visvesvaraya National Institute of Technology (VNIT), Nagpur (India) for providing an opportunity to attend Summer School on CFD during May 26 - July 9, 2016. The present work was executed during the school.

### REFERENCES

1. http://airfoiltools.com/airfoil/details?airfoil=s1091-i1.

2. D.Ravi "CFD prediction on console design for film cooling in gas turbine" 2014, Middle-East Journal of Scientific Research 20 (7): 856-859, ISSN1990-9233.

3. Shridharparegouda, Prof.Dr. T.Nageswararao "CFD simulation on gas turbine blade and effect of hole shape on leading edge film cooling effectiveness" 2013, International Journal of Modern Engineering Research (IJMER), Vol. 3, Issue. 4, pp-2066-2072, ISSN: 2249-6645.

4. Mahmood silieti, Alain j.kassab, Eduardo Divo "Film cooling effectiveness comparision of adiabatic and conjugate heat transfer CFD models" 2009, International Journal of Thermal Sciences 48 (2009) 2237–2248.

5. Harish.R.Sankar, J.P.Vikas, W.Nrisimhendravarun "Experimental and numerical investigation of effect of blowing ratio on film cooling effectiveness and heat transfer coefficient over a gas turbine blade leading edge film cooling configurations" 2013, International Journal of Engineering Research & Technology (IJERT), Vol. 2, Issue 8, ISSN: 2278-0181.

6. Jr-Ming Miao,chen-yuan wn. "Numerical approach to hole shape effect on film cooling effectiveness over flat plate including internal"2005, International Journal of Heat and Mass Transfer 49 (2006) 919–938.