



Effect of RANS-Type Turbulence Models on Adiabatic Film Cooling Effectiveness over a Scaled Up Gas Turbine Blade Leading Edge Surface

Giridhara Babu Yepuri¹ · Ashok Babu Talanki Puttarangasetty² · Deepak Kumar Kolke³ · Felix Jesuraj¹

Received: 28 October 2015 / Accepted: 19 May 2016 / Published online: 23 June 2016
© The Institution of Engineers (India) 2016

Abstract Increasing the gas turbine inlet temperature is one of the key technologies in raising gas turbine engine power output. Film cooling is one of the efficient cooling techniques to cool the hot section components of a gas turbine engines in turn the turbine inlet temperature can be increased. This study aims at investigating the effect of RANS-type turbulence models on adiabatic film cooling effectiveness over a scaled up gas turbine blade leading edge surfaces. For the evaluation, five different two equation RANS-type turbulent models have been taken in consideration, which are available in the ANSYS-Fluent. For this analysis, the gas turbine blade leading edge configuration is generated using Solid Works. The meshing is done using ANSYS-Workbench Mesh and ANSYS-Fluent is used as a solver to solve the flow field. The considered gas turbine blade leading edge model is having five rows of film cooling circular holes, one at stagnation line and the two each on either side of stagnation line at 30° and 60° respectively. Each row has the five holes with the hole diameter of 4 mm, pitch of 21 mm arranged in staggered manner and has the hole injection angle of 30° in span wise direction. The experiments are carried in a subsonic cas-

cade tunnel facility at heat transfer lab of CSIR-National Aerospace Laboratory with a Reynolds number of 1,00,000 based on leading edge diameter. From the Computational Fluid Dynamics (CFD) evaluation it is found that K-ε Realizable model gives more acceptable results with the experimental values, compared to the other considered turbulence models for this type of geometries. Further the CFD evaluated results, using K-ε Realizable model at different blowing ratios are compared with the experimental results.

Keywords Gas turbine blades · Film cooling · Blowing ratio · Density ratio · CFD · RANS turbulent models

List of symbols

RANS	Reynolds averaged Navier–Stokes
SST	Shear stress transport
RNG	Reynolds normalized group
CFD	Computational fluid dynamics
RPT	Rapid prototyping method
B.R	Blowing ratio
D.R	Density ratio
T_c	Coolant temperature, K
T_f	Film temperature, K
T_m	Mainstream temperature, K
T_w	Wall temperature, K
u	Velocity, m/s
v	Velocity, m/s
x	Distance, m
d	Diameter of film hole, m

Greek symbols

η	Adiabatic film cooling effectiveness
ρ	Density, kg/m ³
∞	Mainstream

✉ Giridhara Babu Yepuri
giris@nal.res.in

¹ CSIR-National Aerospace Laboratories, Bangalore 560017, India

² Department of Mechanical Engineering, National Institute of Technology Karnataka, Mangalore 575025, India

³ Department of Mechanical Engineering, University Visvesvaraya College of Engineering, Bangalore 560001, India

Introduction

The efficiency of gas turbine depends highly on the turbine inlet temperature, but the inlet temperature is limited to the allowable temperature of blade material. Film cooling is used in many applications to reduce convective heat transfer to a surface of an object. In order to increase the life of the blade and efficiency, the optimized cooling of gas turbine blade leading edge surfaces is essential.

CFD is a powerful tool for the analysis of geometrically complex engineering problems, although the inclusion of more and more details implies the increasing computational cost. Therefore, a reasonable method must be found, identifying the minimum complexity with the simulation of the main features, as observed in the experiments.

There are many film cooling papers available in the open literature with different geometries. The fundamentals of film cooling and the effect of different variables like surface curvature, freestream turbulence level and length scale, holes shape and configuration and their arrangement on the film cooling performance at different parts of the turbine blade in terms of film effectiveness and heat transfer coefficient was explained by earlier researchers [1]. Turbine airfoil surfaces, shrouds, blades, tips and end walls are cooled by using the discrete hole film cooling.

Multi row film cooling with span wise inclined film cooling holes, called showerhead, is extensively used for cooling the leading edge regions of cooled turbine vanes and blades. Schematic of film cooling is explained by prior investigators [2], as shown in Fig. 1a. Some researchers [3] have presented 3D numerical investigations on the effect of film cooling on the thermal behavior of gas turbine blades, using a commercial CFD code. They considered the two cooling configurations, namely four rows film cooling and eight rows film cooling with U-bend internal channel, have been simulated to be transonic flow over a turbine blade. Some of the investigators have [4] explained detailed film cooling measurements on a turbine blade leading edge model with three rows of showerhead holes. Experiments run at a mainstream Reynolds number of 19,500 based on cylindrical leading edge diameter. One row of holes is located on the stagnation line and the other two rows are located at $\pm 15^\circ$ on either side of the stagnation line. Detailed film effectiveness measurements are obtained using a transient infrared thermography technique. Results show that, the shaping of showerhead holes provides higher effectiveness than the baseline typical leading edge geometry. The researchers have [5] studied the fan-shaped holes, for film-cooling effectiveness and has been numerically analyzed and optimized through 3D RANS analysis

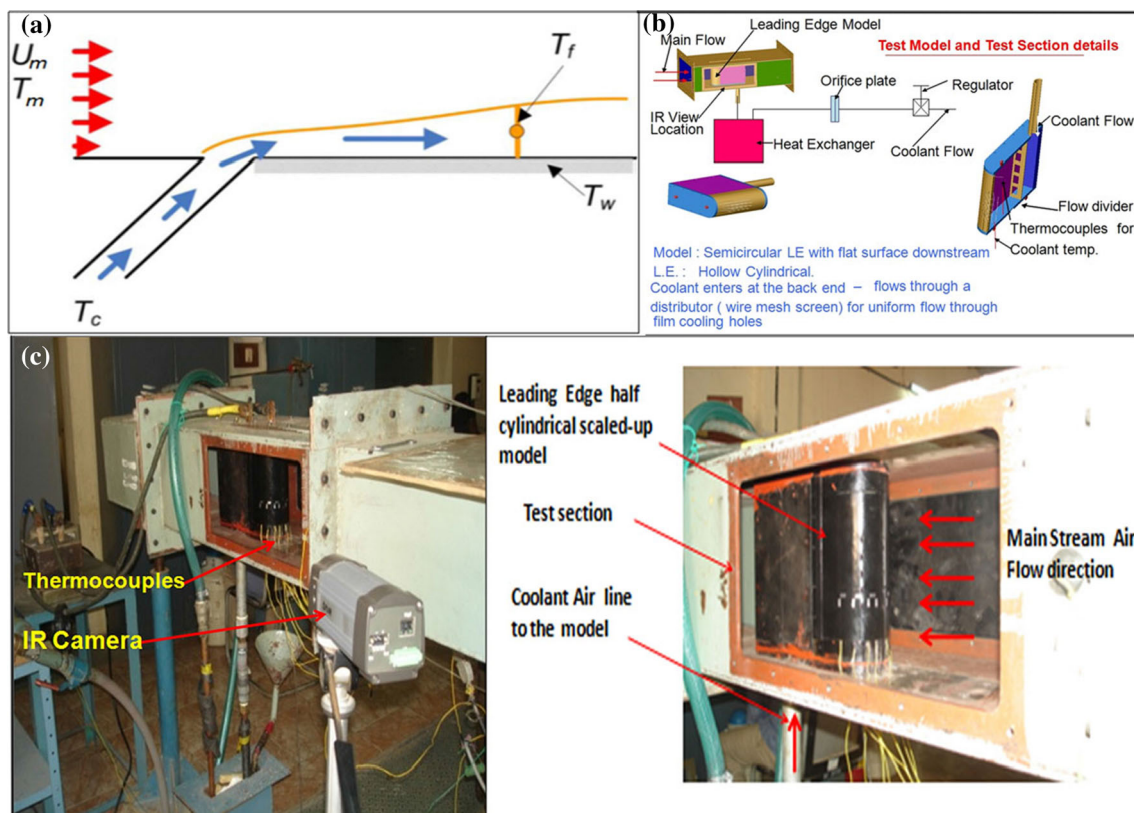


Fig. 1 Schematic and photographic view of film cooling test setup

and weighted-average surrogate model. The literature [6] has shown the comparison of surface heat transfer between a fully 3-D Navier–Stokes code with film injection and the experimental data obtained on a transonic rotating rotor blade with film cooling. Even, some of the researchers have [7] studied the numerical simulation of turbine blade film cooling at different blowing ratios and pitch using $K-\epsilon$ turbulence model showed the temperature field of different location hole rows on leading edge of turbine cascade. The experimental study was made with several test cases with two blowing ratios and three mainstream turbulence intensities using two types of leading edge models with cylindrical holes and diffuser holes [8]. The earlier investigators have [9] studied on the effect of blowing ratio on leading edge film cooling used LES turbulence model to analyze and quantify the effects.

From the above detailed literature survey, it is found that the optimization of turbine blade leading edge film cooling requires the investigation of various flow and geometrical parameters. These parameters include coolant to mainstream blowing ratio, density ratio, the hole inclination angle, hole shape, hole location and the diameter of the hole. By adopting the correct numerical approach based on the type of geometry, the costliest experiments can be avoided. The different authors have applied various grid types, discretization schemes and turbulence models to solve the problems. There are no known numerical studies on evaluating the effect of RANS turbulence models on gas turbine blade leading edge film cooling in the literature. Under the present study, the numerical evaluation is carried out for the optimised two equation RANS turbulent model for the adiabatic film cooling effectiveness over the gas turbine blade leading edge surfaces. A nominal flow Reynolds number of 1,00,000 based on the leading edge diameter, density ratio of 1.3 and blowing ratio of 1.5 are considered during this analysis. And by using the optimised turbulence model, the numerical results are also extracted at other blowing ratios and compared with the experimentally obtained adiabatic film cooling effectiveness results.

Experimental Setup and Procedure

Under this study, a leading edge model with 30° hole injection angle circular holes with five rows of holes is considered for the adiabatic film cooling effectiveness. These five rows of holes are located one at stagnation, two rows of holes at 30° and 60° on either side from the stagnation. Each row is having five holes and the holes are arranged in staggered manner. Gas turbine leading edge model is mounted in the test section consisting of rectangular duct with a size of $320 \times 210 \times 700$ mm. The

experimental test facility consists of compressed air unit, settling chamber, air filter, control valve, orifice meter and rectangular ducts with test section where gas turbine leading edge model is placed. Air is used as a working fluid for both the mainstream and coolant. Mainstream flow is arranged to flow through the settling chamber to have the uniform flow and to the test section. The main flow is controlled by the gate valve placed much ahead of the settling chamber. The cooling air to the model passes through the heat exchanger, where the controlled liquid nitrogen is used to cool the coolant air to have the required coolant temperature. The static and total pressures of mainstream flow to the inlet of test section are measured and maintained to have the required Reynolds number. The coolant flow passing through the orifice meter is also maintained by monitoring the upstream and differential pressures across the orifice meter. Pressure net scanner is used for measuring pressures from pressure ports of coolant and main stream conditions. The required coolant flow is maintained to have the blowing ratios of 1.0, 1.5, 2.0 and 2.5. The mainstream and coolant temperatures are maintained and monitored at the required density ratio of 1.3 using the Fluke data acquisition system.

M/s Flir make A325SC Infrared thermal camera is used for the non contact type temperature measurement of the test surface. The camera offers a high quality, non-intrusive method for obtaining thermal data through a commercially available software package for data analysis. The camera has a capacity to measure the temperature up to 1200°C in three range steps and is having the resolution of 320×240 pixels with the accuracy of temperature measurement $\pm 2^\circ\text{C}$ or $\pm 2\%$ of full scale reading. The test surface is viewed through a thin stretched polyurethane sheet. The sheet is thin enough to cause very little effect on IR transmissivity. Before doing the experiments, the IR system calibrated using thermocouples placed on the black painted test surface. The calibrated K type thermocouple data is captured from the test surface and used to estimate the emissivity of the test surface. The emissivity of the black painted test surface is 0.97. The calibrated transmissivity for the polyurethane sheet and emissivity are checked over a range of temperatures from -20 to 100°C . The effects of atmospheric radiations are considered with the calibration curves. The calibration is in situ and followed the procedure as shown in the literature [10]. In addition to the in situ calibration, the calibrated reference thermocouples are also placed on the leading edge model test surface, to correct the captured IR thermal image data for more accuracy. Figure 1b shows the schematic of experimental test set up and Fig. 1c shows photographic view of experimental setup with infrared camera and data acquisition system. The thermal data is captured for the

considered blowing ratios by setting the relevant test flow conditions at a density ratio of 1.3.

The density ratio is estimated using the relation:

$$\text{Density Ratio} = \rho_c / \rho_\infty$$

The blowing ratio is estimated using the relation:

$$\text{B.R.} = (\rho_c V_c) / (\rho_\infty V_\infty)$$

The mainstream mass flux ($\rho_\infty V_\infty$) is estimated from the measured mainstream velocity and the estimated mainstream density based on the static pressure and total temperature measured in the test section. The coolant jet velocity (V_c) is estimated from the plenum total pressure and the test section static pressure and the coolant density (ρ_c). The average blowing ratio is calculated from the ratio of coolant to main stream mass flux. Adiabatic film cooling effectiveness is found numerically at a density ratio of 1.3 with the coolant to mainstream blowing ratios in the range of 1.0–2.5 at a nominal flow Reynolds number of 1,00,000 based on the leading edge diameter. The local leading edge wall temperature is mixture temperature of the coolant and mainstream the adiabatic film effectiveness is found using the following relation,

$$\text{Film cooling effectiveness } (\eta) = (\mathbf{T}_m - \mathbf{T}_w) / (\mathbf{T}_m - \mathbf{T}_c)$$

The numerically evaluated adiabatic film cooling effectiveness results are compared with the obtained experimental results.

Error Analysis

An experimental uncertainty analysis is followed as per earlier researches [11]. The calculated uncertainty in blowing ratio is ± 0.06 based on the mainstream velocity uncertainty of ± 0.05 m/s, coolant velocity uncertainty of ± 0.6 m/s and the thermocouple accuracy of ± 0.5 °C. The test wall temperatures are captured using infrared camera having the accuracy of ± 2 °C, which makes the uncertainty of ± 5 % on the maximum cooling effectiveness of 0.5. At the lower cooling effectiveness values the uncertainty in the calculated blowing ratio is ± 0.09 based on mainstream velocity uncertainty of ± 0.05 m/s, coolant velocity uncertainty of ± 0.95 m/s which makes the uncertainty of ± 10 % on the minimum cooling effectiveness of 0.08.

Computational Methodology and Procedure

Film cooling effectiveness for blowing ratio of 1.5 with different RANS models for leading edge configuration with circular hole geometry is numerically found using ANSYS-Fluent to have the comparative study with the experiments.

Five different two equation RANS turbulent models such as Standard K- ϵ , RNG K- ϵ , Realizable K- ϵ , Standard K- ϵ and SST K- ϵ turbulence are used to solve the flow field.

Computational Model and Domain Details

The model for computational study is generated using Solid Works. The Fig. 2a shows the gas turbine blade leading edge model with the same dimensions as that of experimental case. Further the computational domain model is prepared in ANSYS-Workbench as per experimental test section dimensions and assuming the symmetry along the length of the test section and model. Figure 2b shows the computational domain with boundary named selection.

Mesh Generation

The ANSYS Workbench mesh is used to create a computational grid. The tetrahedral mesh is constructed for whole domain with inflation layers to capture the boundary layer and suitable wall function is assumed during the mesh generation. Figure 2c shows tetrahedral mesh with inflation on boundary of leading edge. The grid dependency study is performed at a blowing ratio of 1.5 for different mesh sizes i.e. for different element numbers. The averaged mesh quality of all these mesh is found to be higher than 86 %. Figure 2d shows four different meshes with element numbers of 436,198, 661,084, 1,137,648 and 1,435,231.

The grid dependency is plotted for all these considered mesh sizes and is as shown in Fig. 3a. Among the considered mesh sizes, the mesh with 6,61,084 elements and higher showed the same results, hence 6,61,084 elements are considered suitable for further CFD studies.

Boundary Conditions

The types of boundary conditions applied for the computational domain are shown in Table 1. The boundary condition values used in this analysis are same as that of experimental test values. The momentum and energy governing equations are applied in the solution part of the analysis.

Solution setup, Calculations and Post processing

The generated mesh is imported into FLUENT solver where mesh size and quality is checked and confirmed. Solution setup and calculations are done in FLUENT solver, with the unstructured mesh. The different types of boundary conditions applied to the solution are shown in Table 1. For the spatial discretization of all flow variables involved in governing equations, second order methods are selected for the best results as compared to the other methods in FLUENT. Convergence is determined based on

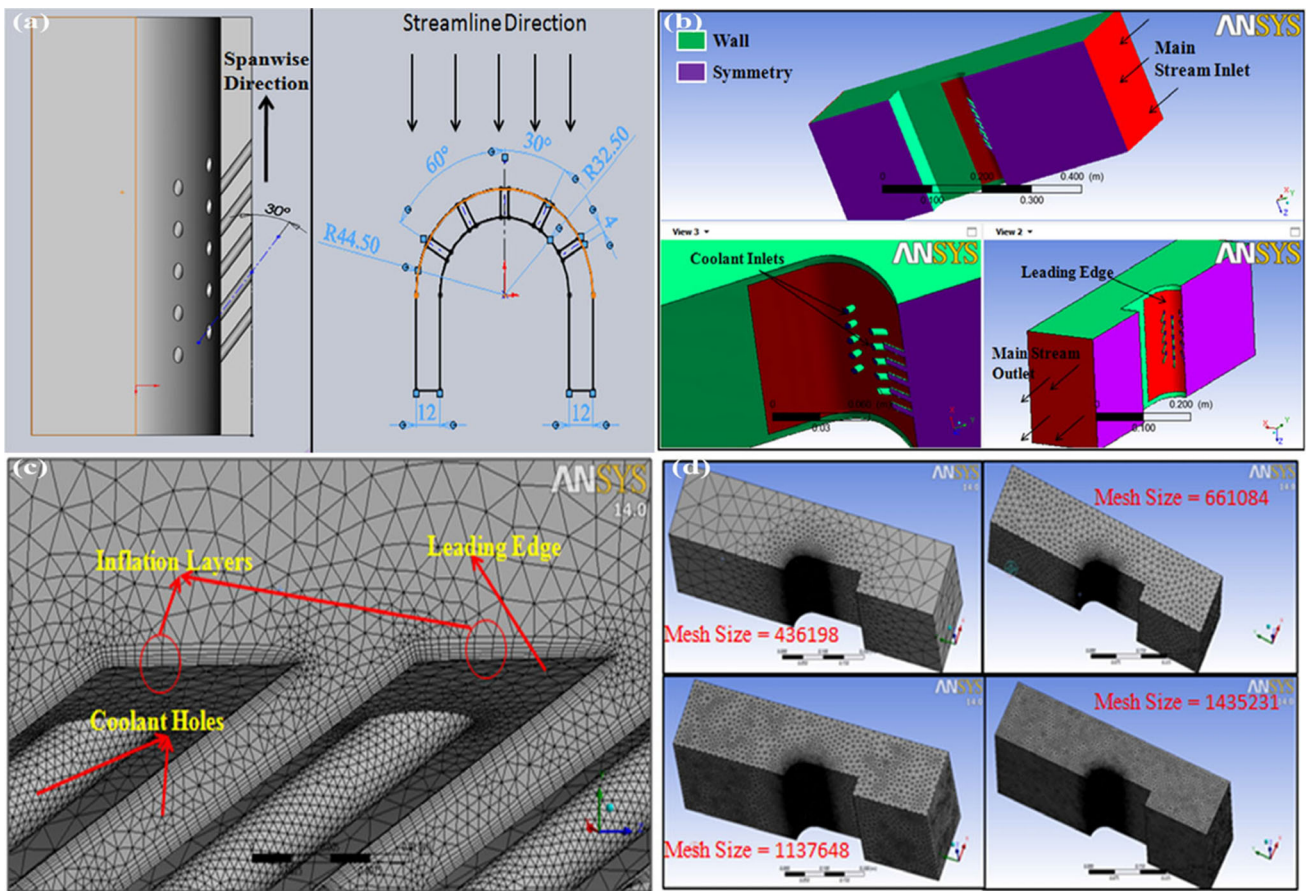


Fig. 2 Geometry, flow domain and mesh sizes of a computational model

Table 1 Boundary conditions

Sl. no.	Surface/part	Boundary type
1	Main stream inlet	Pressure inlet
2	Main stream outlet	Pressure outlet
3	Coolant inlet	Pressure inlet
4	Coolant outlet	Interior
5	Wall	Wall
6	Leading edge	Wall
7	Symmetry	Symmetry

two criteria, one is based on the normalized residual value of absolute convergence criteria and another one by setting surface monitors at mainstream outlet. Post processing is done in CFD Post for the extraction of results.

Results and Discussion

Optimisation of the two equation RANS turbulent models for the adiabatic film cooling effectiveness over the gas turbine blade leading edge surface is carried out. Under this

study, experimental and numerical evaluation of film cooling effectiveness is carried out over the scaled up gas turbine leading edge surface at a density ratio of 1.3 with the coolant to mainstream blowing ratios in the range of 1.0–2.5. Nominal flow Reynolds number of 1,00,000 based on the leading edge diameter is considered in both these experimental and numerical analysis. The film cooling effectiveness results are extracted from the half side of the symmetrical leading edge test model.

Film cooling effectiveness values are estimated by experimental and CFD analysis for considered circular hole geometry over the leading edge of the turbine blade. The spanwise averaged film cooling effectiveness plots in stream wise direction at a blowing ratio of 1.5 are obtained for the five different two equation RANS turbulent models such as Standard K-ε, RNG K-ε, Realizable K-ε, Standard K-ε and SST K-ε turbulence, are shown in Fig. 3b. All these plots are compared with the experimental results at B.R = 1.5. It is observed that among the considered RANS two equation turbulence models, Realizable K-ε model is found to compute the better solution, with a meaningful range compared to that of the experimental values.

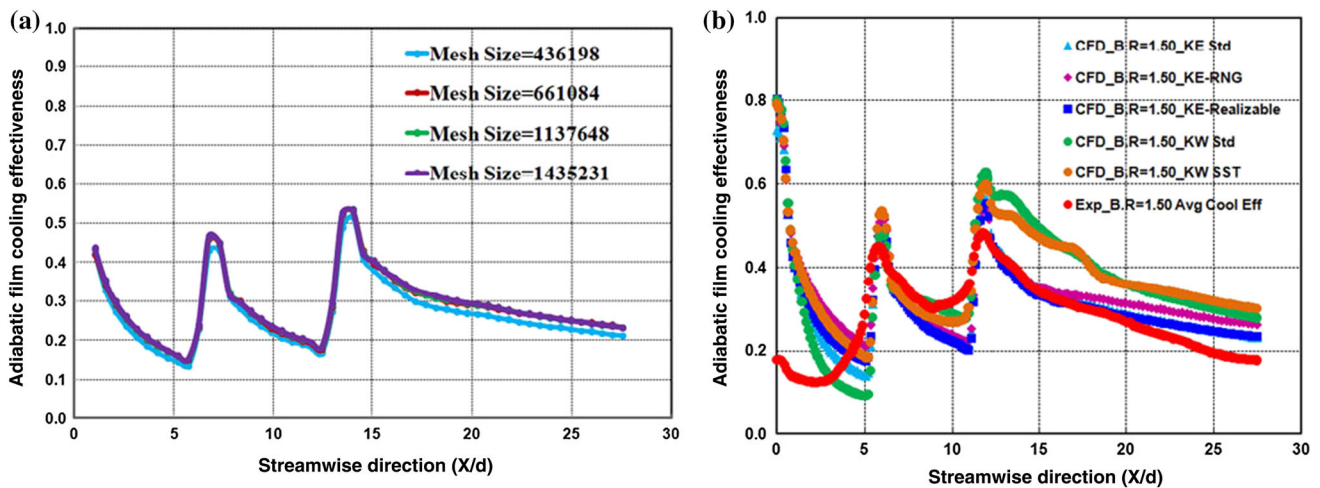


Fig. 3 Comparative plot of optimized mesh size and turbulence models

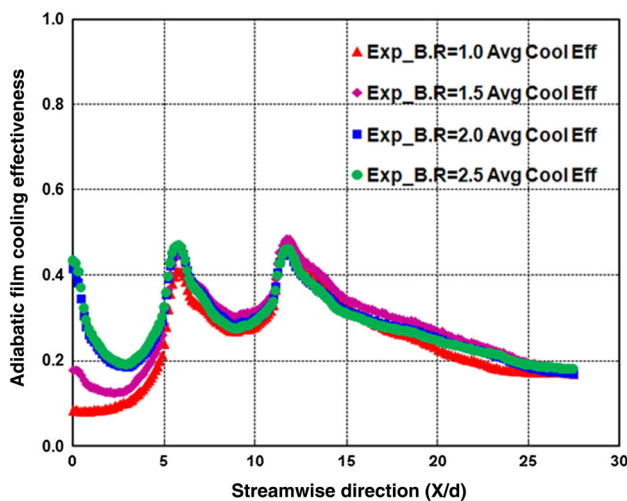


Fig. 4 Experimental averaged adiabatic film cooling effectiveness

The experimentally obtained spanwise averaged film cooling effectiveness plots in stream wise direction at blowing ratios of 1.0, 1.5, 2.0 and 2.5 with the density ratio of 1.3 is shown in Fig. 4.

The numerical and experimental comparative film cooling effectiveness values are shown in the Fig. 5. Figure 5a shows the comparison plot of Realizable $K-\epsilon$ turbulence model with the experimental values at a B.R of 1.5. Similarly, Realizable ($K-\epsilon$) turbulence model is used to find spanwise average film cooling effectiveness values in the stream wise direction at a blowing ratios of 1.0, 2.0 and 2.5, these values are compared with experimental results at the same conditions. Figure 5b–d show the comparative plots of the experimental and CFD film cooling effectiveness results.

From the plots, it is observed that the Realizable ($K-\epsilon$) turbulence model shows the meaningful results with the

similar trends as that of the experimental values. The more deviation of CFD results in the stagnation region is due to the inability of CFD to find the main flow resistance and mixing flow phenomenon accurately at this region. However, at the downstream regions of cooling holes, the Realizable $K-\epsilon$ turbulence model results are well matched with the experimental values. The peaks in the plots have clearly shown the cooling hole row locations in both the experimental and CFD plots. Both the experimental and CFD results have shown the higher cooling effectiveness at a blowing ratio of 2.0 and above that, has not shown any improvement in the cooling effectiveness.

Conclusion

Film cooling effectiveness is found using ANSYS-Fluent similar to experimental conditions to have the comparative study with the experiments. Five different two equation RANS turbulence models such as Standard $K-\epsilon$, RNG $K-\epsilon$, Realizable $K-\epsilon$, Standard $K-\omega$ and SST $K-\omega$ turbulence models are used to solve the flow field.

From this study, it is observed that Realizable $K-\epsilon$ turbulence model has shown the best meaningful results compared to the other considered two equation RANS turbulence models at a blowing ratio of 1.5. The deviation of CFD results in the stagnation region is due to the non predicting of main flow resistance and mixing phenomenon at this region. At the downstream regions of cooling holes, the Realizable $K-\epsilon$ turbulence model results are matched well with the experimental values. Further, the Realizable $K-\epsilon$ turbulent model results are validated by comparing with the experimental values at different blowing ratios of 1.0, 2.0 and 2.5. The comparison plots of CFD and experimental values are in the same trend in an acceptable range.

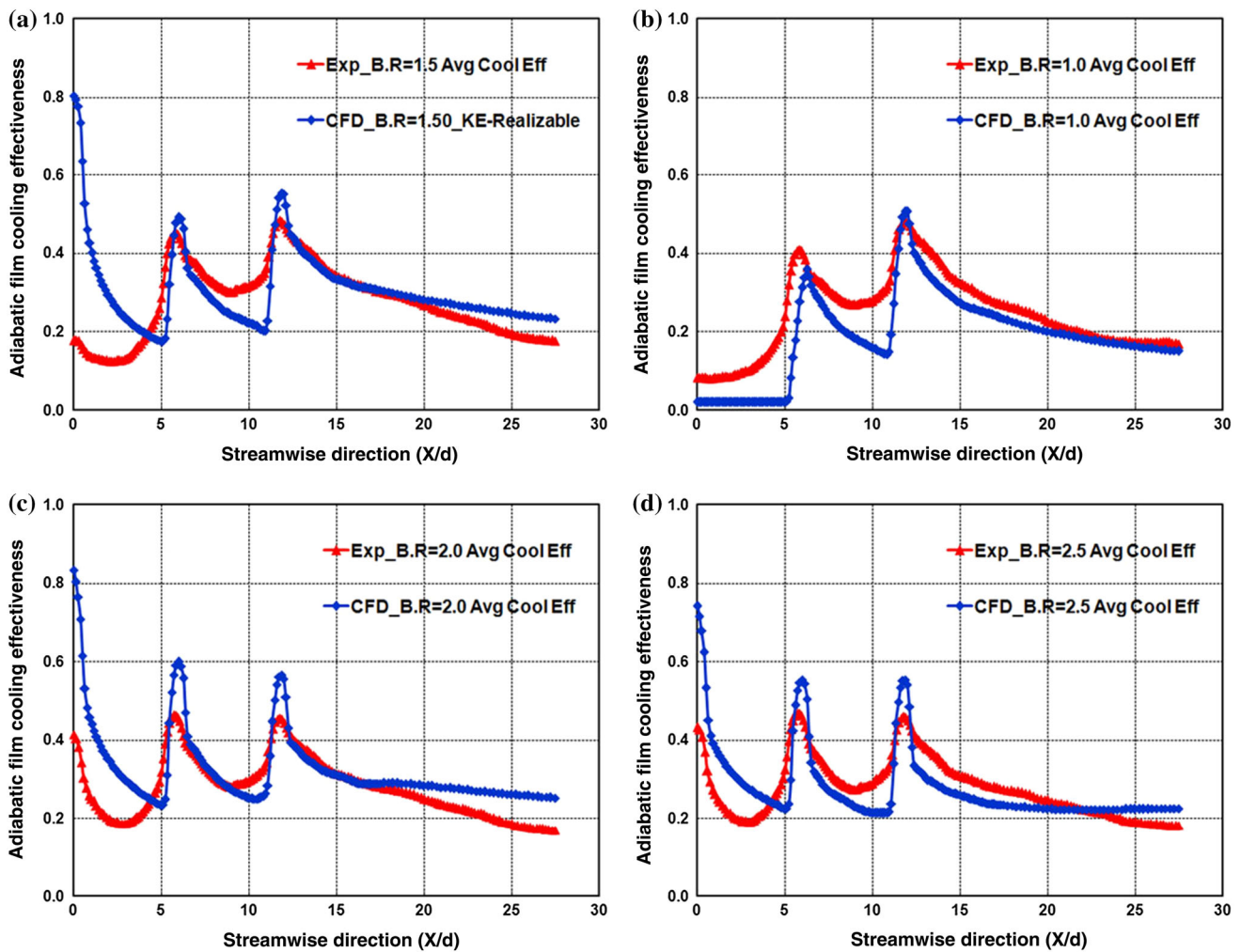


Fig. 5 Experimental against CFD averaged adiabatic film cooling effectiveness

The peaks in the plots have clearly shown the cooling hole row locations in both the experimental and CFD analysis. Both the experimental and CFD results concluded the blowing ratio of 2.0 as an optimized blowing ratio for the considered cooling geometry.

Thus, the Realizable ($K-\epsilon$) turbulence model can be used as a better RANS two equation turbulence model to find the trends of film cooling effectiveness of these types of cooling configurations.

Acknowledgment This paper is a revised and expanded version of the article entitled, “Effect of RANS-Type Turbulence Models on Adiabatic Film Cooling Effectiveness over a Scaled Up Gas Turbine Blade Leading Edge Surface” presented in “2nd National Propulsion Conference” held at Indian Institute of Technology Bombay, Mumbai, India on February 23–24, 2015.

References

1. D.G. Bogard, K.A. Thole, Gas turbine film cooling. *J. Propul. Power* **22**(2), 249–270 (2006)

2. J.E.-C. Han, S. Ekkad, Recent development in turbine blade film cooling. *Int. J. Rotat. Mach.* **7**(1), 21–40 (2001)

3. S.F. Shaker, M.Z. Abdullah, M.A. Mujeebu, K.A. Ahmad, M.K. Abdullah, Study on the effect of number of film cooling rows on the thermal performance of gas turbine blade. *J. Therm. Sci. Technol.* **32**(2), 89–98 (2012)

4. Y. Lu, D. Allison, S.V. Ekkad, Turbine blade showerhead film cooling: influence of hole angle and shaping. *Int. J. Heat Fluid Flow* **28**, 922–931 (2007)

5. K.-D. Lee, K.-Y. Kim, Shape optimization of a fan-shaped hole to enhance film-cooling effectiveness. *Int. J. Heat Mass Transf.* **53**, 2996–3005 (2010)

6. V.K. Garg, R.S. Abhari, Comparison of predicted and experimental Nusselt number for a film-cooled rotating blade. *Int. J. Heat Fluid Flow* **18**, 452–460 (1997)

7. S. Li, T. Peng, L.-X. Liu, T. Guo, B. Yuan, Numerical simulation of turbine blade film cooling with different blowing ratio and hole to hole space, in *International Conference on Power Engineering*, (2007), pp. 1372–1375

8. K. Funazaki, H. Kawabata, Y. Okita, Free-stream turbulence effects on leading edge film cooling. *Int. J. Gas Turbine Propul. Power Syst.* **4**(1), 43–50 (2012)

9. A. Rozati, D.K. Tafti, Effect of coolant–mainstream blowing ratio on leading edge film cooling flow and heat transfer–LES investigation. *Int. J. Heat Fluid Flow* **29**(4), 857–873 (2008)

10. M. Martiny, R. Schiele, M. Gritsch, A. Schulz, S. Wittig, In situ calibration for quantitative infrared thermography, in *QIRT'96 Eurotherm Seminar No. 50*, Stuttgart, Germany (1996)
11. J.P. Holman, *Experimental Methods for Engineers*. McGraw-Hill series in mechanical engineering, 8th edn. (McGraw-Hill, 2011)
12. Y. Giridhara Babu, M. Anbalagan, R. Meena, T.P. Ashok Babu, Experimental and numerical investigation of adiabatic film cooling effectiveness over the compound angled gas turbine blade leading edge model. *Int. J. Mech. Eng. Technol. (IJMET)* **5**(9), 91–100 (2014)