Case Studies in Thermal Engineering 4 (2014) 76–84

Contents lists available at ScienceDirect

Case Studies in Thermal Engineering

journal homepage: www.elsevier.com/locate/csite

Computational investigation of film cooling from cylindrical and row trenched cooling holes near the combustor endwall

Ehsan Kianpour, Nor Azwadi Che Sidik *

Department of Thermo Fluid, Faculty of Mechanical Engineering, University Technology Malysia, 81310 Skudai, Johor, Malaysia

ARTICLE INFO

Article history:
Received 29 April 2014
Received in revised form
15 June 2014
Accepted 10 July 2014
Available online 23 July 2014

Keywords:
Gas turbine
Film-cooling
Cylindrical hole
Trenched hole
Dilution hole

ABSTRACT

This study was performed to investigate the effects of cylindrical and row trenched cooling holes with alignment angles of 0° and 90° at blowing ratio of 3.18 on the film cooling performance adjacent to the endwall surface of a combustor simulator. In this research a three-dimensional representation of Pratt and Whitney gas turbine engine was simulated and analyzed with a commercial finite volume package FLUENT 6.2. The analysis has been carried out with Reynolds-Averaged Navier–Stokes turbulence model (RANS) on internal cooling passages. This combustor simulator was combined with the interaction of two rows of dilution jets, which were staggered in the streamwise direction and aligned in the spanwise direction. Film cooling was placed along the combustor liner walls. In comparison with the baseline case of cooling holes, the application of a row trenched hole near the endwall surface doubled the performance of film cooling effectiveness.

1. Introduction

Advanced gas turbine industries are striving for higher engine efficiencies, and Bryton cycle is the key to achieving this goal. According to this cycle, the turbine inlet temperature should increase to obtain more efficiency. However, the turbine inlet temperature enhancement formed an extremely harsh environment for critical downstream components such as combustor endwall surface and turbine vanes. Therefore, it is essential to plan a cooling scheme in this area. Film cooling is the usual way used. In this system, a thin thermal boundary layer is formed by cooling holes and attached to the protected surface. Cylindrical and trenched cooling holes are the two arrangements of the holes.

Hale et al. [1] measured the effectiveness of surface adiabatic film cooling adjacent to the cooling holes. They noted a variety of L/D ratios, injection angles as well as co-flow and counter-flow plenum feed configurations. The findings of their studies were compatible with the Burd and Simon [2] results which reported that short injection holes enhanced film cooling and created a larger cold area downstream of the cooling holes.

Stitzel and Thole [3] indicated that dilution jet injection is the dominant feature at the combustor exit, while without dilution, the exit profile was relatively uniform with high temperature and low total pressure flow in the mainstream. Furthermore, Scrittore [4] mentioned that increasing the dilution jet velocity adversely affects the surface cooling performance downstream of dilution jets.

Kianpour et al. [5,6] simulated and analyzed the effect of cooling holes geometry at the combustor endwall on the exit flow profiles. The results showed that the temperatures adjacent to the wall and between the jets were about seven percent

* Corresponding author.
E-mail addresses: ekianpour@gmail.com (E. Kianpour), azwadi@fkm.utm.my (N.A. Che Sidik).

http://dx.doi.org/10.1016/j.csite.2014.07.004

cooler with few cooling holes and wider cross-sectional area. In this condition, the coolant spread better near the endwall surface and a wide protected layer was created in order to save the surface against hot gases.

Aga and Abhari [7] and Chien et al. [8] investigated the effects of different inclinations and lateral angles of holes on film cooling effectiveness. They showed that the average adiabatic effectiveness is about twice for streamwise injection at large compound angles (60° and 90°) even for high blowing ratios.

Yiping et al. [9] and Maikell et al. [10] tested the effects of different trench depths and widths on the film cooling under overall cooling effectiveness of $\phi=0.6$. They figured out that $w=2.0D$ and $d=0.75D$ and $w=3.0D$ and $d=0.75D$ cases were more effective than the other cases. For the baseline case, the trench depth of $0.75D$ was optimum and approved by CFD studies.

Azzi and Jurban [11] used various techniques to detect the thermal characteristics of film cooling. In this study, they used standard k–ε turbulence model to solve the Reynolds-Averaged Navier–Stokes equation. In concurrence with Rozati and Danesh [12], the results showed that the film cooling effectiveness was modified at low blowing ratios.

Few researchers [13–16] studied the effects of trenched depth and width on film cooling performance at the vane endwall. Their results showed that the maximum cooling effectiveness is obtained at the trench depth of $0.80D$. However, Lawson and Thole [13] claimed that the trench depth of $0.8D$ has a negative effect on the cooling performance downstream of the cooling hole.

Liang et al. [17,18] conducted few tests to analyze the nonuniformities of temperature adjacent to the wall surfaces and film cooling effectiveness of converging slot hole (console) rows. They concentrated on the effectiveness of the distribution of film cooling using two different exits to inlet area ratios of converging slot hole. The results showed that the coolant injected from the converging slot was attached to the blade surface and enhanced heat transfer.

Somawardhana and Bogard [19] investigated the effects of shallow trenched holes ($h=0.5$ d) to modify film cooling effectiveness. The results were confirmed by Vakil and Thole [20] and Shupping [21] stated that higher film cooling effectiveness for the trenched holes resulted due to the net heat flux reduction that was more than the baseline case, while the heat transfer coefficients for both cases were almost constant.

It appears from the aforementioned investigations that numerous investigations have been conducted for the effects of cooling hole geometry, however, only one attempt was made to investigate the effect of trenching cooling holes near the combustor endwall surface [22,23]. This leads to several unanswered questions: How do trenched cooling holes modify the film cooling performance at the combustor endwall surface in comparison with cylindrical holes? How do cylindrical and trenched cooling holes perform at the same blowing ratios? Therefore, the purpose of the current study was to analyze the film cooling effectiveness variation with different arrangements of cooling holes. Also, in order to identify the validity of the findings, a comparison between the data attained from this investigation and the Vakil and Thole [20] project was carried out.

2. Methods and materials

In this research, a three-dimensional representation of a true Pratt and Whitney gas turbine engine was simulated and analyzed to obtain fundamental data. Fig. 1 shows the schematic view of the combustor. The combustor simulator length, width and inlet height were 156.9 cm, 111.8 cm and 99.1 cm, respectively. The contraction angle was 15.8° and it began at $X=79.8$ cm. While the inlet cross sectional area was 1.11 m^2, the exit cross sectional area was 0.62 m^2. The combustor contained four streamwise film cooling panels. The combustor simulator consisted of a streamwise series of four film-cooled panels, which were symmetrical on the top and bottom of the combustor simulator. These panels began approximately 2.7 vane chords ($\sim$1.57 m) upstream of the turbine test section. The first two panel lengths were 39 and 41 cm while the third and fourth panels were 37 cm and 43 cm, respectively. The panels were 1.27 cm in thickness. Theses panels were made from low thermal conductivity of combustor panels allowed for adiabatic surface temperature measurements.

The second and third cooling panels contained two different rows of dilution holes. These dilution rows were located at 0.67 m and 0.90 m downstream of the beginning of the combustor liner panels. The first and second rows of dilution holes diameter were 8.5 cm and 11.9 cm, respectively.

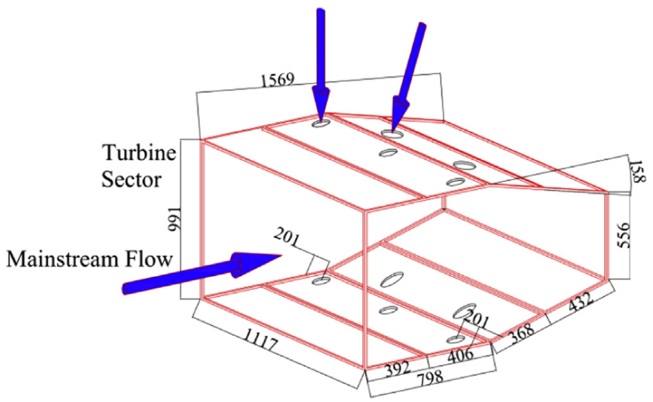

Fig. 1. The 3-D view of the combustor simulator.