International Journal of Heat and Mass Transfer 61 (2013) 389–396

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

Thermodynamic analysis of flow field at the end of combustor simulator

Ehsan Kianpour [a], Nor Azwadi Che Sidik [a,*], Mohsen Agha Seyyed Mirza Bozorg [b]

[a] Department of Thermo Fluid, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM, Skudai, Johor, Malaysia
[b] Iranian Space Agency (ISA), Tehran, Iran

ARTICLE INFO

Article history:
Received 14 July 2012
Received in revised form 4 February 2013
Accepted 11 February 2013
Available online 5 March 2013

Keywords:
Thermal contour
Gas turbine
Dilution hole
Film-cooling
Cooling hole

ABSTRACT

This study was accomplished in order to investigate the effects of different cooling hole configurations on the thermal and flow field characteristics inside a combustor simulator. By using the well-known Brayton cycle, great turbine industries try to extend the inlet temperature and augment engine performance. However the turbine inlet temperature increment creates an extremely harsh environment for the downstream components of the combustor. In this research a three-dimensional representation of Pratt and Whitney aero-engine was simulated and analysed with a commercial finite volume package FLUENT 6.2 to gain fundamental data. The current study has been performed with Reynolds-averaged Navier–Stokes turbulence model (RANS) on internal cooling passages. This combustor simulator combined the interaction of two rows of dilution jets, which were staggered in the streamwise direction and aligned in the span wise direction, with that of film-cooling along the combustor liner walls. The entire findings of the study declare that the greater penetration depth, the thicker the film-cooling layers. Furthermore, in the combustor simulator with more cooling holes, the temperature near the wall and between the jets was slightly increased. Also at the leading edge of the jet, and at the jet–mainstream interface, the gradients of temperature were quite high.

1. Introduction

Modern gas turbine industries strive for higher engine efficiencies. Brayton cycle is a key to this study. According to this cycle, the turbine inlet temperature should increase to gain more efficiency. However increasing the turbine inlet temperature creates an extremely harsh environment for critical downstream components such as turbine vanes. Fuel must be thoroughly mixed with air, completely burned, and made to flow uniformly before entering the turbine. However, insignificant mixing leads to non-uniformities such as hot streaks and allows non-combusted fuel to exit the combustor. Hot streaks can cause premature wear and ruin turbine components. In the present study, a broad literature search was conducted to collect the information and specify the combustor characteristics.

Kwak and Han [1] analysed the effects of Mach number and blowing ratios on heat transfer coefficient by using transient liquid crystal technique. The results declare that cooling blockage leads to blow rate enhancement; as a result, decreases in tip heat transfer coefficient and increases in static pressure above the shroud are found.

By using large scale, low speed experiments, Rowbury et al. [2,3] determined the effects of flow interaction under the annular cascade application. They investigated the effects of holegeometry, pressure ratio across the hole, Reynolds number and Mach number on discharge coefficient of coolant. The results indicate that near the end of an injection hole with external cross flow, the static pressure loss relative to the assumed value led to the discharge coefficient enhancement.

By using a test set, Zhang and Chang [4] determined the effect of spares placed inside cooling holes on blade heat transfer. Like a previous study by Herbert [5], it was observed that the spare film cooling hole application facilitated an effective heat transfer with little impingement in comparison with pure initial cross flow.

Yuzhen et al. [6] determined the film cooling effectiveness of three different Multihole schemes. The findings indicate that for the multihole pattern, a slight spanwise pitch is desirable. Optimum cooling is attained by using a combination of different hole schemes.

Hale et al. [7] 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 Burd and Simon [8] results which reported that short injection holes enhanced film cooling and created a larger cold area downstream of the cooling holes.

Tarchi et al. [9] were able to identify the effects of a large dilution hole which was placed within penetrating slot and effusion array. In accord with Milanes et al. [10], it can be seen that by using

* Corresponding author. Tel.: +60 7 5534718; fax: +60 7 5566159.
E-mail address: azwadi@fkm.utm.my (N.A.C. Sidik).

0017-9310/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.02.031

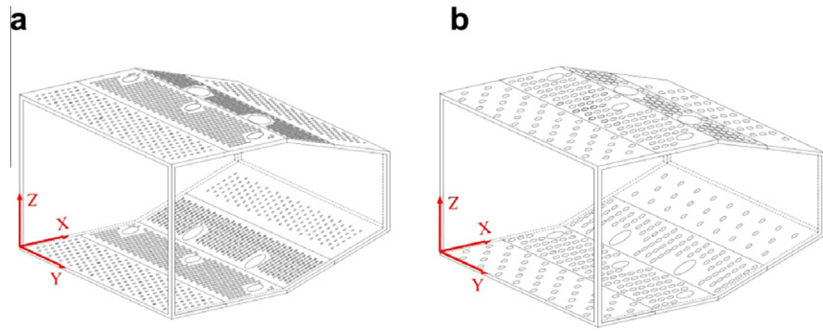

Fig. 1. Schematic view of the combustor simulator (a) Case 1 (b) Case 2.

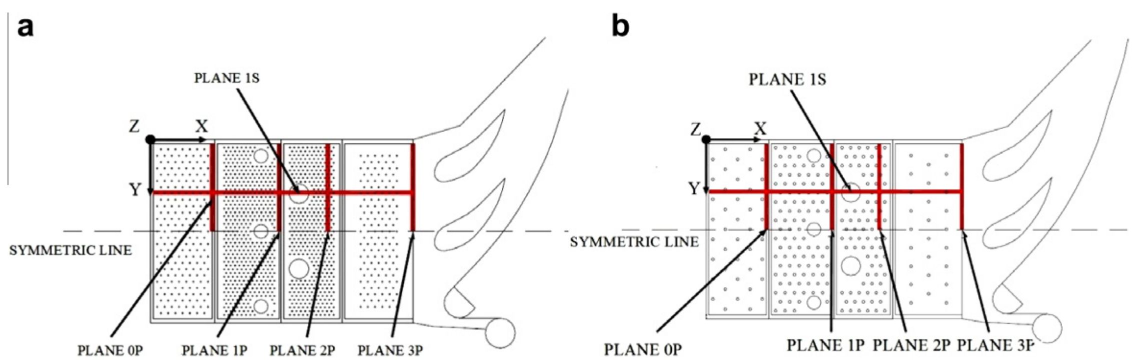

Fig. 2. Location of the measured planes (a) Case 1 (b) Case 2.

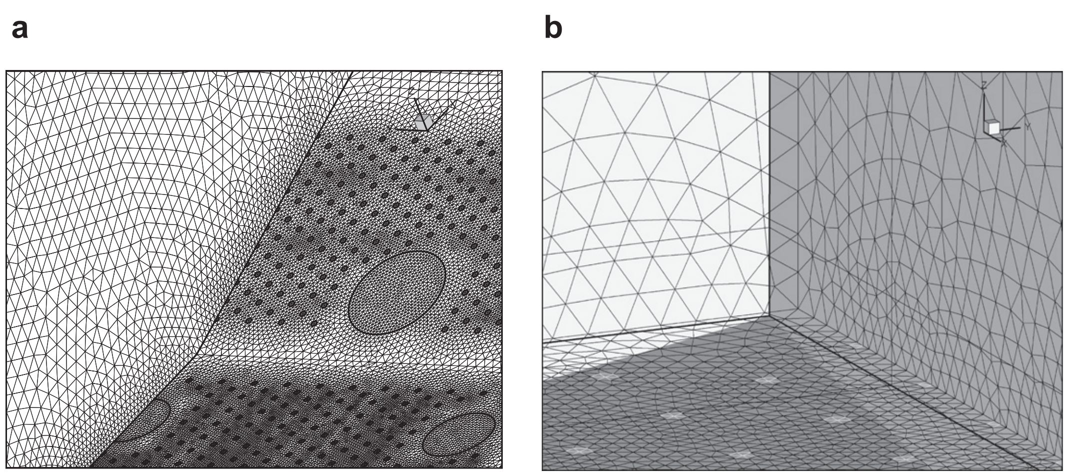

Fig. 3. Three-dimensional meshes of combustor simulator (a) Case 1 (b) Case 2.

the backward step, the dilution hole affected the adiabatic film cooling effectiveness downstream and reached to $\eta_{aw} = 0.65$.

Stitzel and Thole [11] indicated that dilution jet injection is the dominant feature at the combustor exit, while with no dilution, the exit profile was relatively uniform with a high temperature and low total pressure flow in the mainstream. Furthermore Scrittore [12] mentioned that increasing the dilution jet velocity adversely effects the surface cooling performance downstream of dilution jets.

Cun Liang et al. [13,14] conducted an experimental investigation to study the non-uniformities of temperature near the wall surfaces and film cooling performance of converging slot hole (Console) rows. They focused on the effectiveness of distribution of film cooling using two different exits to inlet area ratios of converging slot hole. The results show that the coolant injected from the converging slot attached to the blade surface and enhanced heat transfer.

Lu [15] and Zhong and Brown [16] studied the performance of different hole patterns and blowing ratios for a row of cylindrical holes. Variations of hole patterns differed in stream wise row spacing, and span wise hole pitch and hole inclination angle were tested under two different test rigs. Results show that while downstream rows are protected by stream wise coolant injection, reverse coolant injection provides superior cooling protection for initial rows.

Saumweber et al. [17] prepared a test section to study the distribution of free stream turbulence under different shaped holes. The results show that for the cylindrical holes, heat transfer vari-