Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 04, April-2018

Numerical Analysis of a Multiholed Flat Plate for Gas Turbine Combustor Liner Application

Alosri Prajwal
M Tech Student
Aeronautical Department, MVJCE
Bangalore Karnataka, India

Kumar Gaurav
Assistant Professor
Aeronautical Department, MVJCE
Bangalore Karnataka, India

Rampada Rana
Scientist 'E'
GTRE, DRDO
Bangalore Karnataka, India

***Abstract*— A multi-hole cooling method seems a promising cooling technology for combustor liner protection. Compared with traditional methods it has higher cooling effectiveness, which can decrease the cooling mass flow requirements and possibly allow changes in combustor geometry. The process of reducing the cooling flow is done using a number of small holes and the higher effectiveness is due to higher temperature drop of the plate because of combined effect of convective cooling through the holes and additional film cooling effectiveness. Numerical study of effusion cooling flow and heat transfer analysis is conducted using a Reynolds-averaged Navier–Stokes (RANS) approach. Realizable k-e turbulence model with enhanced wall treatment is used to predict the flow field of an array of 10 rows of effusion holes, each hole inclined at 30° to the flat plate. Computational averaged cooling effectiveness across the plate is predicted and compared with experimental available results of a multi-hole plate.**

Keywords*— *Cooling Effectiveness; Multiholed Plate; Effusion Cooling

NOMENCLATURE

CFD	Computational Fluid Dynamics
CHT	Conjugate Heat Transfer
T_h	Mainstream Temperature
T_c	Coolant Temperature
T_p	Flat Plate Temperature
η	Cooling Effectiveness
p	Transverse Pitch
d	Hole Diameter
S	Longitudinal Pitch
α	Inclination Angle

I. INTRODUCTION

Gas turbine engines operate at very high temperatures, thus, components such as turbine end walls, combustion chamber walls, shrouds, and turbine blades need to be cooled. With higher thrust to weight ratio (specific thrust) and the desire to increase the thermal efficiency of gas turbines by reducing the specific fuel consumption, has led to an increase in temperature and pressure in the combustion chamber and turbine. The operational life of the combustion chamber walls decreases as the temperature increases [1]

To satisfy the increase of both turbine entry temperatures and the compressor outlet, the properties of metal-alloys operated in gas turbine components and the techniques for cooling them must be improved [2]. Improvement in material properties can be least expected, apart from the improvement of high-temperature ceramic components. There is a requirement of technology for cooling metal walls efficiently, thus, a technique for cooling must be employed to protect the wall. Almost all current gas turbine engines employ the film cooling method for combustor chamber wall cooling.

Among these cooling techniques effusion cooling (multi-hole) is extensively employed in modern engines because of its simplicity and less mass flow rate requirement. In this technique, cold air is injected through small holes in the chamber wall which creates a film of cooler fluid and hence acts as a barrier between the hot gases and wall material [3]. It is also found that the combustion liner has been protected from hot combustion gas using forced convectional cooling methods such as rib roughened surface and jet impingement cooling [4]. The estimation of liner temperature though CHT analysis is also found to be carried out in many cases considering the reactive flow.

In the present work, temperature distribution in multiholed flat plate is numerically predicted using commercially available CFD software, ANSYS Fluent 17. The objectives of the present investigation are (1) to obtain an accurate temperature distribution and cooling effectiveness on the multiholed plate; and (2) to compare CFD results with available experimental results by B. Leger, P. Miron, and J.M. Emidio [2].

II. COMPUTATIONAL METHODOLOGY

Computational fluid dynamics is a branch of fluid mechanics which uses numerical analysis and data structures to solve and analyze problems that involve fluid flows and heat transfer. It solves Navier–Stokes equations to simulate the fluid flow and heat transfer problem considering conservation of mass, momentum and energy equation.

A. *Geometric details*

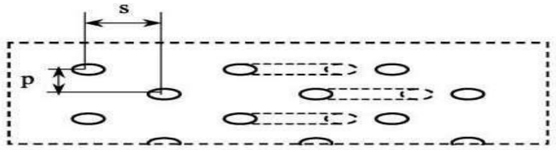

Fig 1: Geometrical configuration of multiholed plate top view.

The work carried out by B. Leger, P. Miron, J.M. Emidio [2] focuses on obtaining cooling effectiveness on multiholed plate under various of geometric and aero thermal parameters. The effusion plate, comprised of cylindrical cooling holes inclined at 30° and arranged in both the stream wise and span wise directions with staggered pattern, is illustrated in Fig.1 (top view) and Fig.2 (front view).

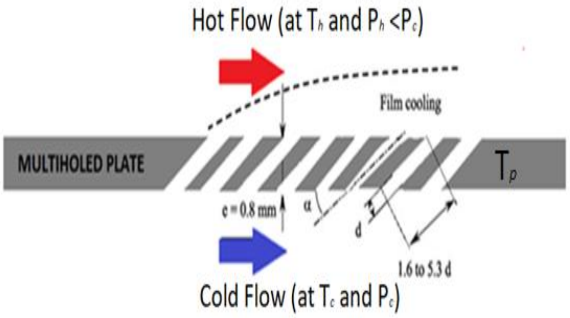

Fig 2: Geometrical configuration of multiholed plate front view

To carry out the numerical analysis, the plate has been modeled in CAD software, fluid extraction, geometry cleaning is performed in Space Claim. A computational domain of hot fluid of height 70 mm and cold fluid of height 50mm was extended vertically above and below of the multiholed plate respectively for uniform flow conditions as showed in Fig.3 Geometry and specification of plate was maintained as in reference paper [2]

B. *Grid details*

The accuracy and correctness of CFD analysis depends on the quality of grid. Simulations were performed on three different grid sizes of 6 million, 8 million and 11 million elements respectively to assess the sensitivity of the mesh against predicted results. Fig.4 shows the meshed geometry along with the computational domain.

To capture the heat transfer and pressure loss effectively, it is required to capture the boundary layer at the grid level. The geometry meshed such that the multiholed plate and the regions near the multiholed plate had a relatively finer mesh as compared to the computational domain. The grid near to the wall have been resolved by carrying out initial analysis and subsequent estimation of boundary layer by capturing initial flow profiles of the system in terms of density, viscosity (fluid properties) and velocity (flow properties) for the length (geometrical property). The grid consists of 11 million elements with the hexahedral generated in ANSYS 17. First layer thickness being 0.0126mm is generated to capture the boundary layer. Fluent guide [4] is used to generate the grid.

III. SOLVER DETAILS

For the present analysis ANSYS Fluent 17.0 is used.

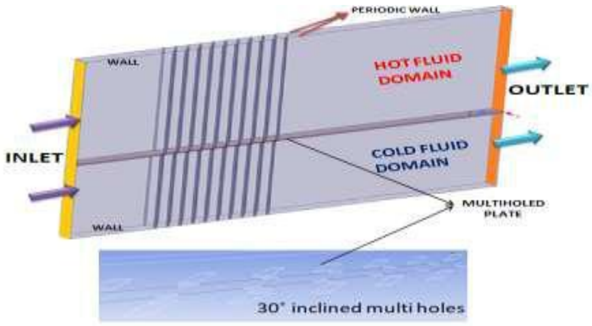

Fig 3: computational domain of multiholed plate.

Analysis is carried out considering the flow as a three dimensional, steady state, incompressible, turbulent, flow coupled with heat transfer.

A. *Turbulence model.*

In the present analysis, the two-equation Realizable k-ε turbulence model with enhanced wall treatment is used. The simulation was run using a second-order upwind scheme.

B. *Boundary conditions*

Inlet and Outlet are taken as mass flow inlet and pressure outlet respectively. The operating condition of the experimental setup [2] is considered in the present computation. Temperature for hot and cold domain is 1161K and 300K respectively. Mass flow rates of hot and cold domain have been estimated as 0.008355kg/s and 0.018255 kg/s respectively. Material properties of steel used for the analysis is given in the Table 1

Table 1: Material properties of air and steel

Material	Fluid: Air	Plate: steel
Density(kg/m3)	1.225	7850
Specific Heat Capacity(J/kg.K)	1006.43	490
Thermal conductivity (W/m-K)	0.0242	50.2

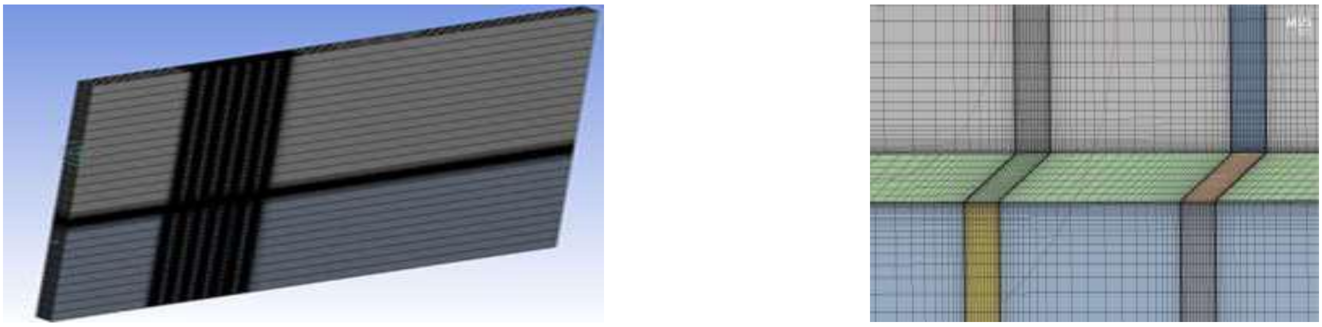

Fig 4: View of multiholed plate computational domain grid

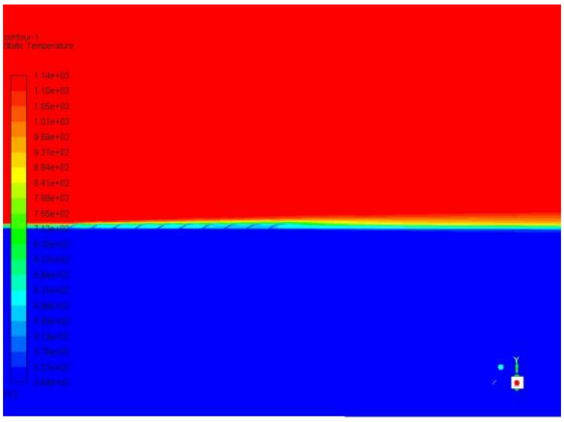

Fig 5: Static temperature contour of the flow field at mid plane

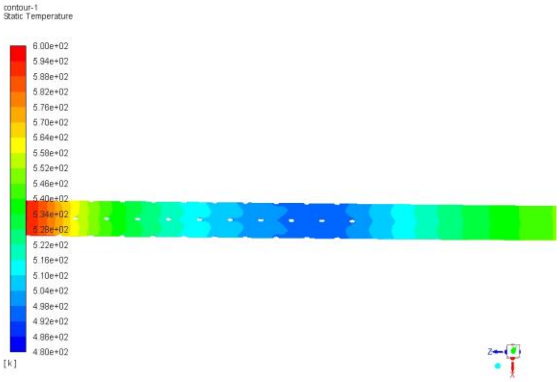

Fig.6: Static temperature contour of the multiholed plate

IV. RESULTS AND DISCUSSION

The results of CFD analysis of multiholed plate are detailed in this section. Temperature profile on the flat plate, variation of velocity and effusion cooling effectiveness has been estimated. Cooling effectiveness was estimated on the effusion test plate by using equation (1).

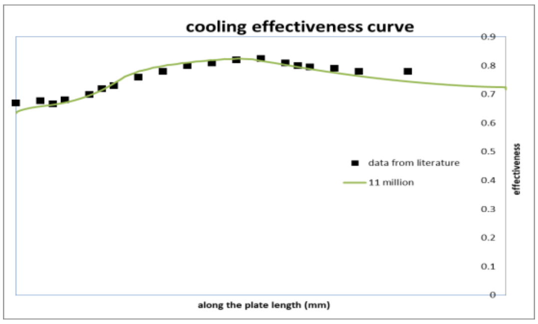

Fig.7: comparison of estimated η and experimental data

A. *Comparison between experimental data and the CFD analysis*

Fig.7 shows the CFD values compared with experimental value. The estimated values at almost all data points were quite close to the experimental data. The mean error of predicted data was estimated to be approximately 5%. Cooling effectiveness η [5] a dimensionless wall temperature parameter defined as

$$\eta = (T_h - T_p) / (T_h - T_c) \qquad (1)$$

Fig.5 shows the static temperature variation in the flow field taken at the mid plane. The contour shows the increment of temperature on the flat plate due to the hot flow upstream of the multiholed region and decrease in temperature near the multiholed region where maximum effusion cooling effectiveness is achieved downstream of the multihole, it was observed that the multiholed zone of the plate leads to a large decrease in plate temperature. This decreasing temperature effect is due to the increasing convective heat transfer inside the holes and also, to the creation of a film cooling effect on the hot side of the plate as in the Fig 6

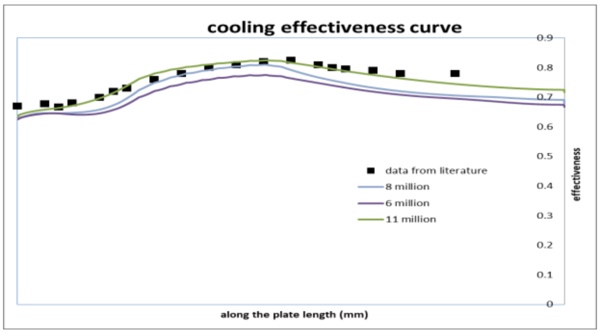

Fig.8: grid independent study

A comparison of the effectiveness plots of the grids has been illustrated Fig.8, the grid with 8 million and 6 million elements is relatively coarser, it was observed that the simulation was not able to capture the subtle variations in temperature (10 to 20 %).

C. Velocity contour at the mid plane Equations

Fig .9 shows the flowing of cold fluid through the cooling holes, the formation of cooling film along the plate.

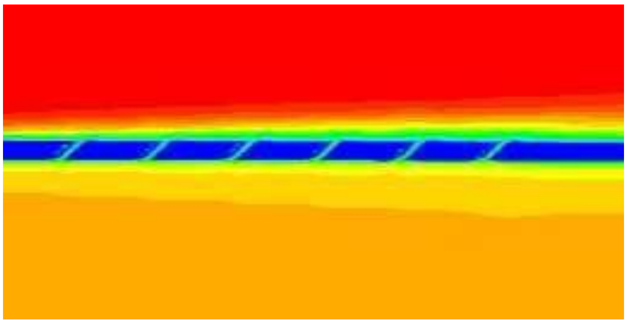

Fig.9: velocity contour at the mid plane

IV. USING THE TEMPLATE

This paper describes a CFD study of effusion cooling flow and heat transfer. Averaged cooling effectiveness follows the experimental measurements reasonably well. The overall trend, i.e. the initial decrease, followed by a gradual increase in effectiveness is predicted. Temperature measure along the flat plate is predicted using CFD fluent tool and the computational temperature range had a close agreement with experimental results.

ACKNOWLEDGMENT

The authors would like to thank Mr. M. Z. Siddique, outstanding scientist and Director, GTRE, DRDO for permitting us to carry out the project work at. Authors would also like to thank Prof. Gupta, HOD, Aeronautical Engineering Department, MVJCE, Bangalore for his valuable support and guidance throughout the project.

REFERENCES

[1] M. Walton and Z. Yang, “Numerical Study Of Effusion Cooling Flowand Heat Transfer”, International Journal of Computational Methods and Experimental Measurements , Volume 2 (2014), Issue 4

[2] B. Leger , P. Miron , J.M. Emidio , Geometric and aero-thermal influences on multiholed plate temperature: application on combustor wall, International Journal of Heat and Mass Transfer 46 (2003) 1215–1222

[3] Reaz Hasann, Agin Puthukkudi, “Numerical study of effusion cooling on an adiabatic flat plate”, Propulsion and Power Research 2013;2(4):269–275

[4] G Manoj Kumar and J Bruce Ralphin Rose “Numerical Comparative Study on Convective Heat Transfer Coefficient in a Combustor Liner of Gas Turbine with Coating” International Journal of Mechanical Engineering and Research, ISSN 0973-4562 Vol. 5 No.1 (2015)

[5] FLUENT 6.1 User's Guide

[6] Nijo Jose, Tayakumar J S, Giridhara Babu Yepuri, “Numerical Investigation of adiabatic Film cooling effectiveness over a flat plate model with cylindrical holes” International Journal of Heat and Mass Transfer (2015) 398-404.