ISSN:2278 – 7798
International Journal of Science, Engineering and Technology Research (IJSETR)
Volume 5, Issue 7, July 2016

Numerical Analysis of Combustor Flame Tube Cooling

SK.MD.Azharuddin, Sabin Adhikari, Syed Imtiaz, Manjunath S.V

Abstract - **Numerical analysis was conducted to understand the combustion of Jet aviation gas in aircraft's combustor. The study was aimed to conducted thermo-fluid analysis of Turbo-Union RB199 Engine combustor and the approach has been made using the application of CFD tool Ansys 15. As combustion process is the complex mechanism involving the principles of thermodynamics, fluid mechanics and the chemical kinetics, the experimental burdens were alleviated by CFD analysis. The single combustor has been designed as a replica of one in the Turbo-Union RB199 Engine, with suitable boundary conditions, temperature and velocity profiles are obtained. Moreover, with the film cooling and effusion cooling methods for proper combustion and maintaining temperature gradient level required by turbine blades, the approach has been made to understand heat transfer and flow pattern in the combustor.**

Index Terms—**CFD, Combustor, Flow Analysis, Thermal Analysis, Heat Transfer, Jet Engine, Numerical method, Film Cooling, Effusion Cooling, Combustor CAD model,**

I. INTRODUCTION

Effective cooling of combustor flame tube has been main area of concern during design and manufacturing of aircraft's engine. Approach has been made to design the combustion chamber for cooling and analyze combustion process, temperature gradient, flow velocity, flame propagation and efficient cooling. Combustion of compressed air and fuel mixture result in high temperature, which may damage the combustor flame tube material and the turbine blade, when this hot gas pass to turbine for expansion. The metal liners of gas turbine engine combustors are provided with some form of thermal protection from the high temperatures of the reacting mixture of gases generated during combustion. The compressed air is introduced by tangential injection as a discrete film at a number of axial stations along the combustor liner.

Manjunath S V, Assistant professor, Department of Aerospace Engineering, Alliance University/, Bengaluru, India

SK MD Azharuddin,, *Student,Department of Aerospace Engineering, Alliance University/ (mazharuddin@ced.alliance.edu.in). Bengaluru, India*

Sabin Adhikari, *Student, Department of Aerospace Engineering, Alliance University/ (asabin@ced.alliance.edu.in). Bengaluru, India*

Syed Imtiaz, *Student, Department of Aerospace Engineering, Alliance University/ (isyed@ced.alliance.edu.in). Bengaluru, India.*

The liner is embodied with cooling slots such that as the cooling potential of one film is depleted it is periodically renewed by another. These slots are called cooling holes whose main function is to act as a relatively cool barrier between the vulnerable liner and the reacting gases. Failure to design cooling slots can result in thermal damage to the liner. The design and operational development of gas turbine combustor is a complex process involving a great volume of design and experimental work. In this project, combustion process has been studied and corresponding cooling effect has been analyzed using the application of computational fluid dynamics (CFD) method.

Design and analysis of can type combustor was carried out by Ch. Umamaheshwar Praveen, [3]. The combustor was designed using the principle of combustor sizing such that all operating points (i.e., idle, full power) are considered and the smallest size combustor geometry was obtained that provide stability over the entire range of operation is chosen.CFD software can be used to carry out the thermo fluid analysis of combustion chamber. The burdens of experimental design and number of practical approaches can be overcome by effectiveness of the CFD tools. Various attempts have been made to analyse combustion chamber using CFD. Crocker and Smith, Snyder et al utilized CFD tools to determine temperature distributions inside combustion chamber by studying the diameter(size) and location of the dilution holes. Moreover, influence of holes on dilution zone to exit temperature was analysed by them [4].

The wall temperature and the cooling effectiveness of deflection hole on effusion cooling were analysed by Liu X et. al, [5], to investigate the effusion cooling performance in real combustion chamber with strong rotation and primary holes provided. Effusion cooling performance was found better than the conventional film cooling methods. With deflection hole on effusion cooling, the wall temperature, gradient was lowered by significant value, the coolant was reduced by 20%, moreover, higher cooling efficiency was obtained. 60 degree deflection of cooling holes was found to be best suited for effective cooling.

The mechanics of film cooling was described by Eidon L. Knuth [6].The effect of high turbulent gas streams on the thin liquid wall films of the combustor were studied by him. The methods for calculating maximum allowable coolant flow rate for stable coolant film, determining the evaporation rate and the surface temperature for stable inert coolant film was found. Similarly, the condition required to ensure liquid film attachment to solid surfaces without the loss of un-evaporated liquid was obtained.

II. OVERVIEW OF TURBO-UNION RB199 ENGINE

The Turbo-Union RB199 is a Turbofan engine used in PanaviaTonado aircraft. It was designed and built in the early 1970s by Turbo-Union, a joint venture between Rolls-Royce, MTU and FiatAvio.

The RB199 has 3-stage low pressure, 3-stage intermediate pressure and 6-stage high pressure compressor. Turbine section is single stage high pressure, single stage intermediate pressure and 2-stage low pressure. Overall pressure ratio of the engine is 23.5:1.

The combustor used in RB199 is Tuboannular type, where there are 10 tubular liners arranged inside a single annular casing. The length of the combustor is 560 millimeters and diameter is of 140 millimeters.

Figure 1: Cut-section of combustor (ACED, Propulsion Lab)

III. GEOMETRIC MODEL

From our literature survey we were not able to find the exact geometry (dimension) of the combustor; even internet source couldn't help us. So we decided to take the model of combustor from jet engine (Turbo-union RB199) present in our college propulsion lab. We took the dimensions of that combustion chamber to make the design in cad software. The dimensions were taken with the help measuring instruments like vernier caliper, micro meter screw gauge, measuring tape etc.

Figure 2: Combustor data collection (ACED, Propulsion Lab)

Figure 3: Combustor data collection (ACED, Propulsion Lab)

Turbo Union RB-199 Rolls RoyceEngine combustor is taken as the Combustor geometry for the study purpose. The CAD design is made in SolidWorks 2015 and CFD analysis was carried out in Ansys Workbench 15.

Figure 4: CAD model of combustor

Figure 4 shows the combustor arrangement in Turbo-Union RB199 engine. This engine having tuboannular combustor consists of 10 combustor assembly inside the engine casing. The combustors are arranged between engine shaft and casing in circular pattern. Each combustor is connected to the adjacent combustor through an interconnector. Some of compressed air enters each and every combustor to participate in combustion while rest of it flows between the liner and engine casing for cooling of flame tube and combusted gases.

Bottom of the figure 4shows the single combustor out of 10 combustor assembly of Turbo-Union RB199 engine. The combustion process and compressed air flow is almost same in all the combustor so approach has been made to take one combustor for study and find the equivalent result suitable for all 10 combustor assembly.

IV. GOVERNING EQUATION [7]

a) Mass conservation equation

$$\frac{\partial \rho}{\partial t} + \nabla . (\rho \vec{v}) = S_m$$

b) Momentum conservation equation

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla . (\rho \vec{v}\vec{v}) = -\nabla p + \nabla . (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F}$$

c) Energy equation

$$\frac{\partial (\rho E)}{\partial t} + \nabla . [\vec{v}(\rho E + p)] = \nabla . \left[k_{eff} \nabla T - \sum_i h_i \vec{J}_i + \tau_{eff} . \vec{v} \right] + S_h$$

i = 1, 2, 3 . . . . is the number of species

Standard k-εModel equation (2 equations) [7]

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_i}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k$$

and

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C_{3\epsilon}G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon$$

V. BOUNDARY CONDITIONS

i. Solver: Pressure based steady state
ii. Viscous Model: Standard k-ϵ, Standard wall function
iii. Radiation Model- P1
iv. Air Inlet Velocity: 140 m/s (M=0.4)
Temperature: 550K
v. Fuel Inlet Velocity: 8 m/s
vi. Outlet: Pressure constant
vii. Wall
Motion - Stationary
Shear Condition - No Slip

Material Used: Nimonic-75
Fuel species used: Jet A

VI. RESULTS

i. Flow analysis

Analysis was carried out in Ansys Fulent 15 to check the combustion and flow of the combusted gas; these results were imported to Ansys Thermal 15 to verify the body temperature of the combustion chamber.The combustor was modeled in SolidWorks 2015 as shown in figure 4.

Velocity Profile

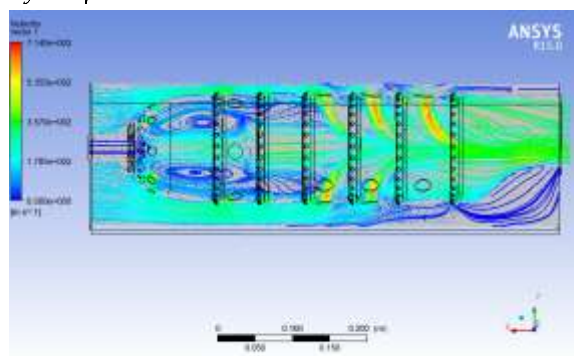

Figure 5: Velocity Stream line

Fuel enters the combustor with velocity of 8m/s as shown in figure 5-36 and the velocity of air from the compressor outlet is about 140m/s. In primary zone the velocity is minimum about 70m/s because of swirling effect and recirculation. Velocity of air in primary zone is reduced from 140m/s to 70m/s this reduction is necessary form proper mixing of fuel and air and as well for combustion. Velocity is reduced with the help of swirler which gives rise to torodial flow pattern in primary zone as shown in figure 5-37, which is essential for continuous ignition.

Temperature Profile

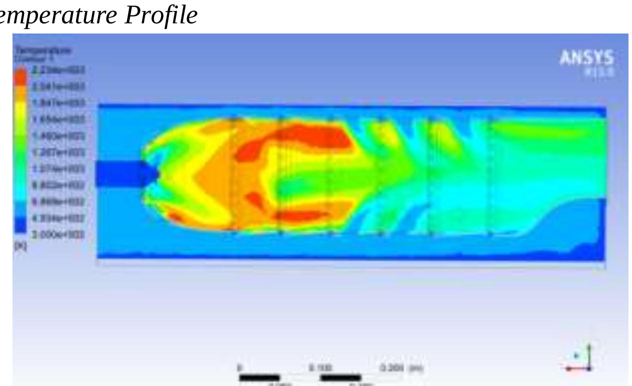

Figure 6: Temperature distribution

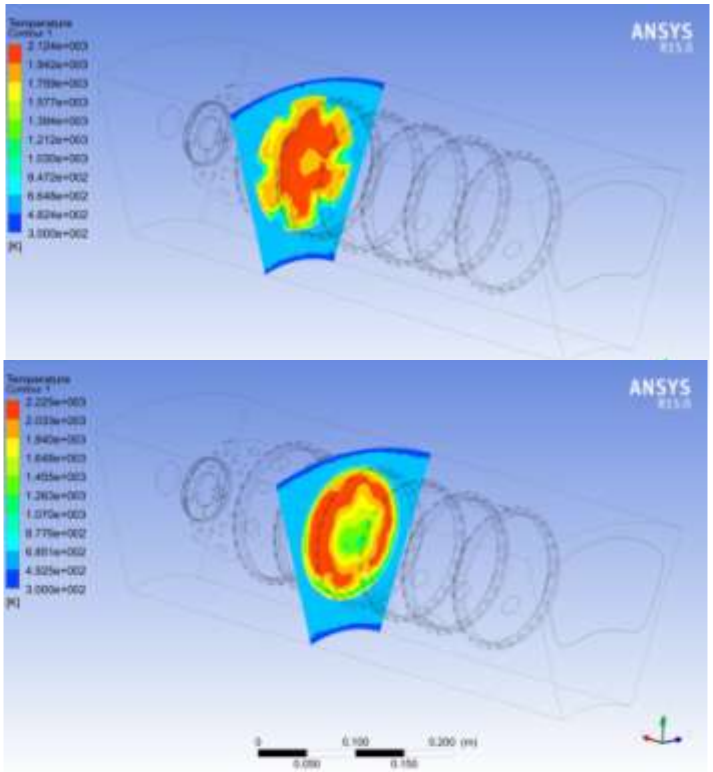

Figure 7: Primary Zone(top) and Intermediate Zone(bottom)

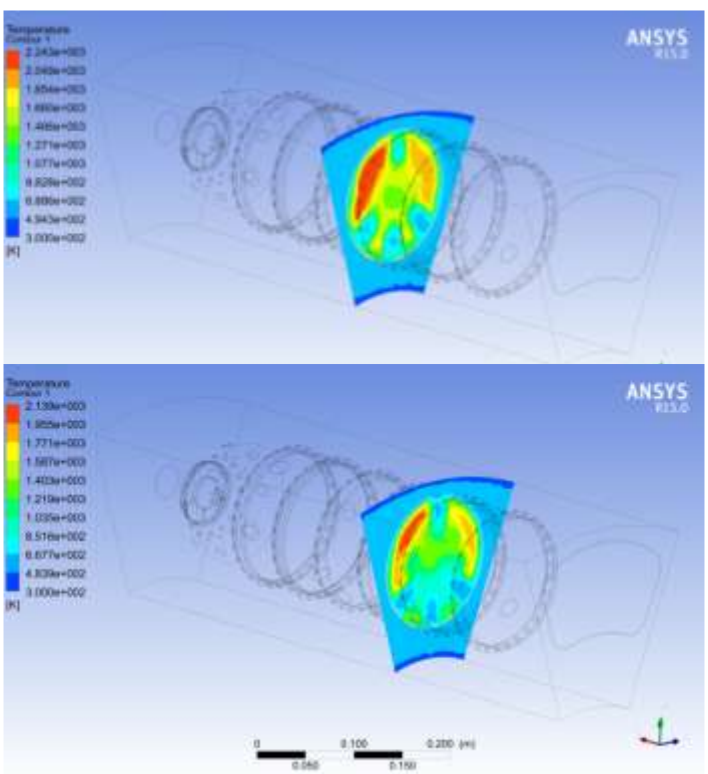

Figure 8: Secondary holes(top) and Dilution section1(bottom)

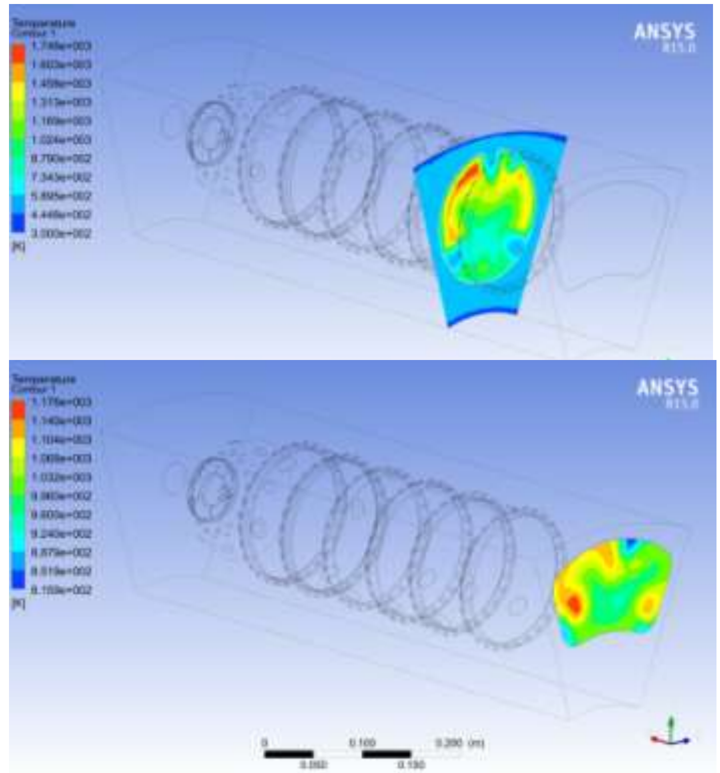

Figure 9: Dilution section2(top) and Outlet temperature(bottom)

Figure 6 shows the temperature distribution along the combustion chamber. It can be observed that fuel is entering through fuel inlet with temperature of 300K, whereas compressed air from compressor is entering at temperature of 550K. Compressed air passes through swirler where recirculation takes place and fuel is sprayed to the incoming air. From figure 5-38 we can see that there is combustion taking place in primary zone where maximum combustion temperature is reaching to 2234K. There are holes in primary zone where air is added. Extra amount of air is added so that complete combustion takes place. In intermediate zone there are no holes because air-fuel mixture should burn completely before reaching dilution zone. From figure 6 we can observe that in intermediate zone complete combustion is taking place. As combusted gas passes to dilution zone it must be cooled before it reaches turbine blade. Figure 8 we can notice that there is cooling taking place where temperature is dropping from 2230K to around 1200K near outlet. In dilution zone holes are placed at different angle and the diameter of the holes varies with section in the dilution zone. Holes are placed at different angle because to cool that part of the region inside combustion chamber. Figure 6 shows the temperature profile in the different section of dilution zone.

As the combusted gas passes to dilution zone more amount of air is added to cool the combusted gas. In figure 8 it can be seen that there is significant change in the temperature as the air is added to combusted gas. In dilution zone section 1(figure 8) there is a hole in the top, which is of diameter 26mm and bottom there is 3 holes which is of diameter 21mm. In section 2 of dilution zone (figure 9) there are total 4 holes, top 2 holes are of diameter 26mm and 12mm and bottom 2 holes are of diameter 21mm.

ii. Thermal Analysis

Steady-State Thermal analysis is utilized to carry out the thermal analysis and to determine temperature distribution on combustor surface. A Steady-State Thermal analysis calculates the effects of steady thermal loads on a system due to heat transfer phenomena.

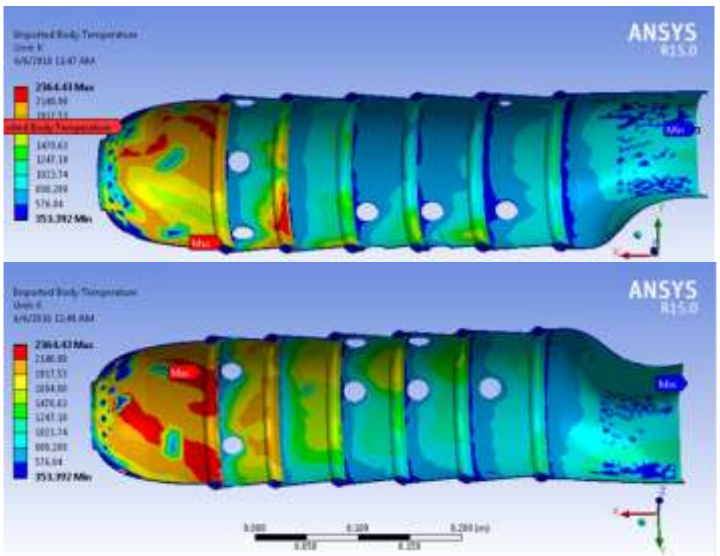

Figure 10: Surface temperature

VII. CONCLUSION

The thermo-fluid analysis of the combustor is carried out using the CFD tools. The velocity and the temperature profile for the combustor are obtained for the given boundary conditions. The flow velocity was minimum in primary zone due to swirling effect but reverse is the case for temperature profile as temperature is maximum in primary zone and decreases towards the outlet due to cooling effect of cooling slots provided on the combustor walls. Hence, influence of cooling slots in the combustion process, flame propagation, flow pattern and orientations, and heat transfer in combustor was realized and its significances were studied.

REFERENCE

[1] “The Jet Engine” Rolls Royce plc, pp. 35-36, 1996.

[2] Arthur H. Lefebvre and Dilip R. Ballal “Gas Turbine Combustion Alternative Fuels and Emissions” CRC press, Taylor and Francis Group, pp.10-17, 140-142, 221-285, 315-356, 2010.

[3] Ch. Umamaheshwar Praveen, ”Design and Analysis of Can Combustor”, Publication in” International Journal of Emerging Technology and Advanced Engineering, Volume 4, Issue 9, September 2014.

[4] D. S. Crocker, D. Nickolaus, C. E. Smith, CFD Modeling of a Gas Turbine Combustor From Compressor Exit to Turbine Inlet, Journal of Engineering for Gas Turbines and Power, JANUARY 1999, Vol. 121 / 89, ASME, pp. 89-95.

[5] Xiao LIU and Hongtao ZHENG,” Influence of Deflection Hole Angle on Effusion Cooling in a Real Combustion Chamber condition”, Publication in” Thermal Science: Year 2015, Vol. 19, No. 2”

[6] Eidon L. Knuth,”The Mechanics of Film Cooling”,Ph D thesis, California Institute of Technology, California, 1954.

[7] Ansys Fluent 12.0 Tutorial Guide; Release 12.0, April 2009.