View metadata, citation and similar papers at core.ac.uk
brought to you by CORE
provided by International Journal of Research and Engineering

To study the influence of cooling holes in a combustor using CFD

Author(s): SK.MD.Azharuddin, Sabin Adhikari
Affiliation: Department of Aerospace Engineering,
Alliance University, Bengaluru, Karnataka, India

Abstract – combustion process is the complex phenomena involving the principles of fluid dynamics, thermodynamics and the chemical kinetics, which occurs in the limited space available inside the combustor. Moreover, proper combustion is the essence to enhance overall combustor efficiency. Combustor liner is provided with the cooling slots i.e. Film cooling and effusion cooling, to accomplice complete combustion and giving optimum temperature gradient required by the turbine blade. These cooling procedures will protect vulnerable combustor liner from hot combusted gas. In order to evince the effect of cooling slots, approach has been made to analyze combustor geometry using the application of CFD tools.

Keywords – **Combustion; Combustor; CFD; Effusion Cooling; Film Cooling; Flow Analysis; Gas Turbine; Heat Transfer; Thermal Analysis;**

I. INTRODUCTION

The main requirements of the combustor is to get high-combustion efficiency (i.e., the fuel should burn completely) along with low pressure loss and outlet temperature distribution that is suitable for turbine blades. These requirements can be achieved by proper cooling in the combustor. In a typical can type combustor 80% [1] of air is used for cooling of combustor liner and combusted gas whereas 12% of air is passed through the swirling vanes to participate in combustion and rest 8% is utilized to cool the dome surface of the primary zone in the combustor.

As the combustion temperature in the combustor reaches to 1600 – 2000℃ it is difficult to carry out the experimental analysis of the combustor. So, to avoid burden of experimental work CFD tools Ansys 15 has been used.

Increase in compressor pressure ratio and turbine inlet temperature enhances thermal efficiency of a gas turbine but it induces the thermal stress in the combustion chamber. So the need for effective cooling becomes more and more vivid.

Therefore, Cooling is necessary for proper combustion and to ensure a satisfactory liner life of the combustor and to maintain the gas temperature as required by turbine operation. Following cooling methods are used in combustor:

- Film Cooling
  This type of cooling is employed to cool the liner of the combustor. The liner is provided with a protective film of cooling air between the wall and the hot combustion gases. The main advantage of the method is that the cooling slots can be designed to withstand severe thermal stresses at high temperatures and pressure.

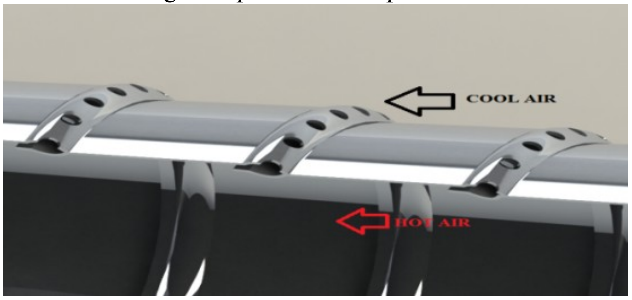

Figure 1: Film Cooling

- Effusion Cooling
  A wall perforated by a large number of holes as shown in figure 2 is called as effusion cooling. The effectiveness is due to increase in convection inside the holes. The convection heat transfer process depends on the heat transfer coefficient h, surface inside the holes and air flow temperature. By optimizing the geometry of combustion chamber, air velocity and efficient cooling of hot gases can be achieved with this method with minimum mass flow rate of air.

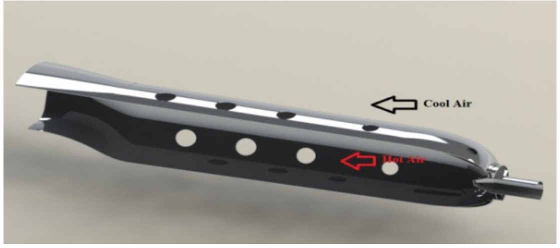

Figure 2: Effusion Cooling

There are plenty numbers of examples of cooling analysis of combustor using CFD solutions in the literature of which are as follows: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 [9].

The wall temperature and the cooling effectiveness of deflection hole on effusion cooling were analysed by Liu X et. al, [3], 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 [4].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.

MajaMunktell [5], explain about the combustion chamber cooling in his report. Higher compressor ratio imparts higher heat transfer to combustor's liner. So, cooling is needed in the combustion chamber. It was concluded that only film cooling is wastage of cooling air mass flow rate, both film and multi-holed cooling should be provided to achieve cooling effectiveness in combustion chamber

Leger B et. Al, [6] made an experimental study of geometric and aero-thermal influences on multi-holed plate temperature on combustion walls. Studies shows that wall temperature decreases from 640K to 440 K. Temperature reductions occurred due to presence of 50mm long multi-holed zone.

Mcguirk and Palma [7], made the study of flow inside the model gas turbine combustor. The flow profile inside the combustor with or without the presence of swirler was found out by them using the applications of CFD. Taking k-e model as the turbulent model inside the combustion chamber, the correct shape of the turbulence kinetic energy distribution across the combustor was visualized.

Effusion cooling holes cool down the hot gases temperature inside the combustion chamber by significant values. H. I. Oguntade et al [8] explained about the influence of number of holes on effusion cooling effectiveness. G and N being massflow rate and the holes per square meter of surface area respectively, the cooling effectiveness of Nimonic-75 material was studied by varying the values of G and N.

II. DESIGN AND ANALYSIS

Geometric Model: Turbo-Union RB-199 Rolls Royce Engine combustor dimension with over length of 560mm and diameter of 144mm is taken as Combustor model for the study.. The 2D diagram of the combustor is shown in figure 3.

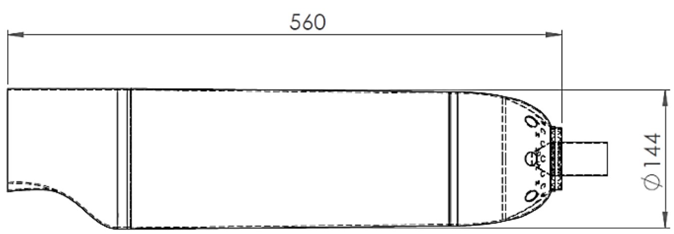

Figure 3: 2D diagram of Combustor

Turbo-Union RB-199 Engine combustor without any cooling methods, with film cooling only and with effusion cooling only, are taken as the design model and corresponding analysis is carried out to visualize cooling slots effectiveness . The model is created in SolidWorks 15 and imported to Ansys 15 for analysis.

Governing equation:

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

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

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

Standard k-ε Model equation (2 equations) [12]

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

and

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

Boundary Conditions

i. Solver: Pressure based steady state
ii. Viscous Model: Standard k-e, 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

III. RESULTS

i. Flow analysis

Model without any cooling

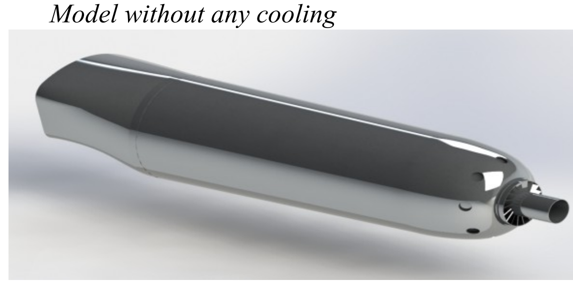

Figure 4: Combustor without Cooling

Hollow cylindrical body is taken as combustor without any holes in the wall as shown in the figure 4.

Velocity Profile

- The velocity is maximum about 1000m/s in primary zone (figure 5) and toward its walls, because of back flow from the wall and recirculation.
- Velocity decreases axially in intermediate and dilution zone with approx. magnitude of 500m/s (figure 5).
- It increases in nozzle guide vane giving outlet velocity of approx. 700m/s (figure 5).

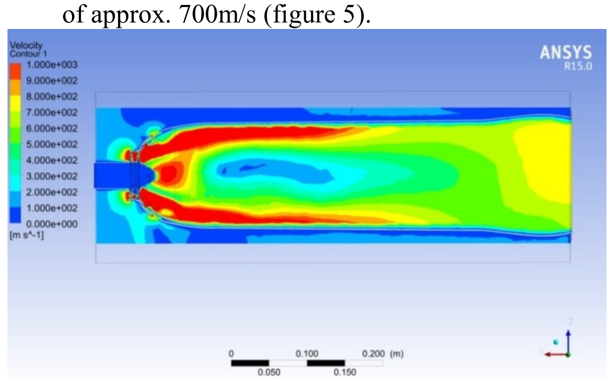

Figure 5: Velocity Contour

Temperature Profile

- Flame is getting stabilized in secondary zone which is undesirable.
- Flame is not uniformly distributed across the section.
- Combustion temperature at Primary zone is about 2200K (figure 6). Since, none of cooling methods i.e. effusion and film cooling methods are applied, same temperature propagates along axis of combustion chamber giving higher outlet temperature of 1230K (figure 7).

- Combustion takes place centrally, so the heat transfer towards the wall is less, maintaining temperature of about 900K (figure 7) near the wall.

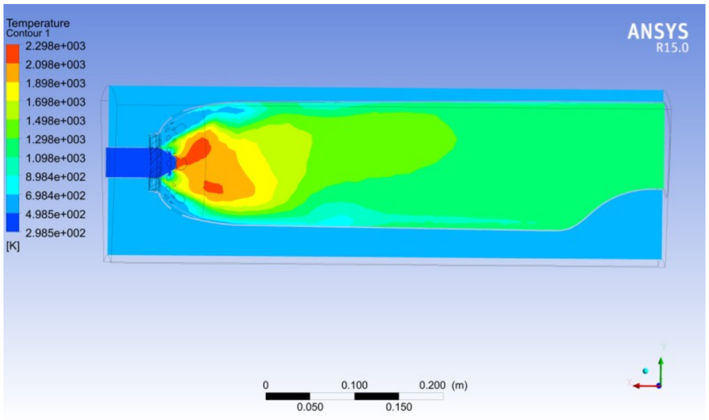

Figure 6: Temperature Contour XY plane

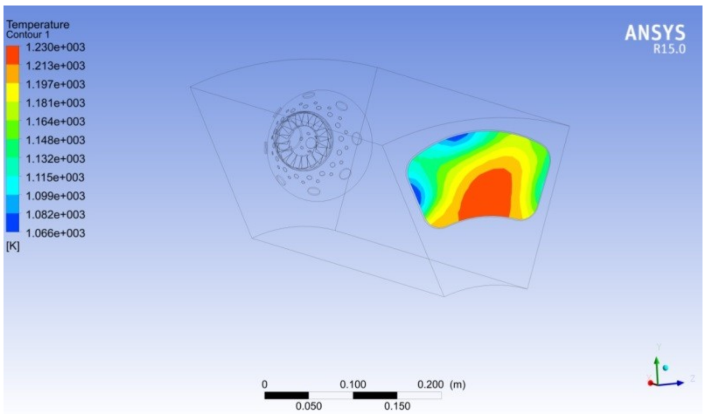

Figure 7: Outlet temperature

Model with Film Cooling

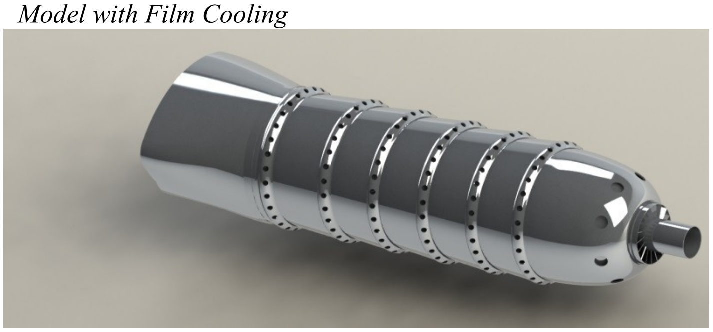

Figure 8: Combustor with Film Cooling

Film cooling to the combustor model is introduced. This method aimed at cooling the surface temperature of the combustor. The velocity and temperature profile for this case are explained below.

Velocity Profile

- The velocity is minimum about 120m/s in primary zone (figure 9) because of swirling effect. This velocity is higher than that of standard model because of back flow from the wall.
- Once the combustion takes place, velocity increases along the length of chamber with approx. 450m/s at dilution zone and approx. 650m/s at outlet (figure 9).

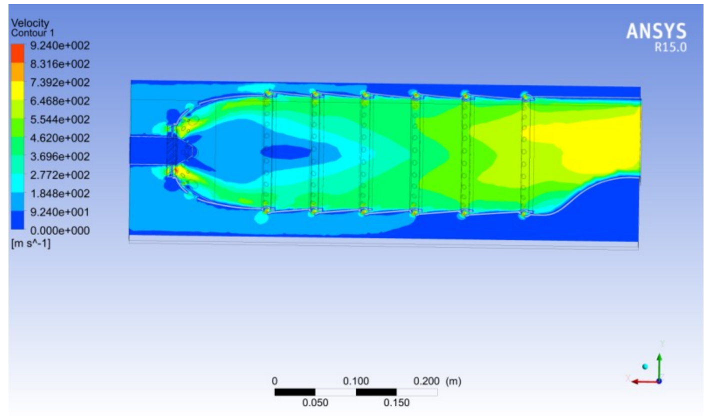

Figure 9: Velocity Contour

Temperature Profile:

- Flame is getting stabilized in secondary zone or dilution zone which is undesirable.
- Flame is uniformly distributed across the section.
- Combustion temperature at Primary zone is about 2200K (figure 10). Since, the geometry isn't provided with any effusion cooling, the flame propagates axially with outlet temperature of approx. 1950K at the centre.
- Temperature distribution towards the wall region is less.

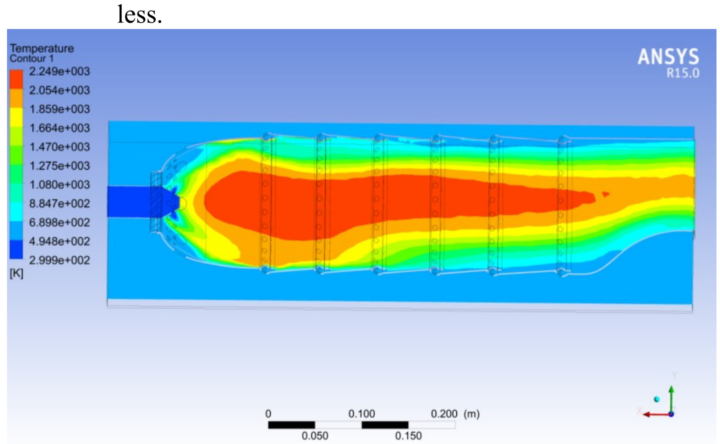

Figure 10: Temperature Contour XY Plane

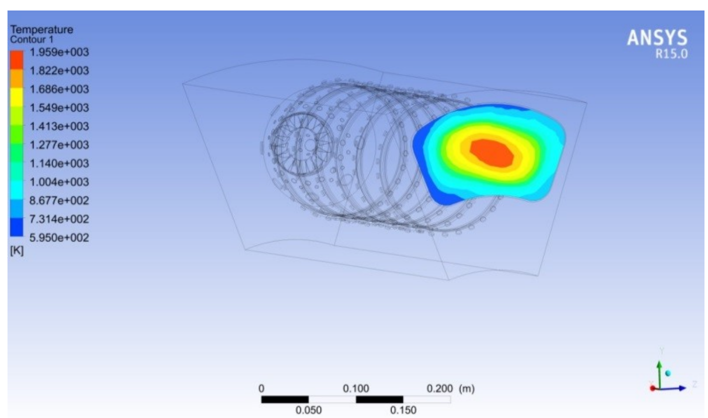

Figure 11: Outlet Temperature

Model with Effusion Cooling

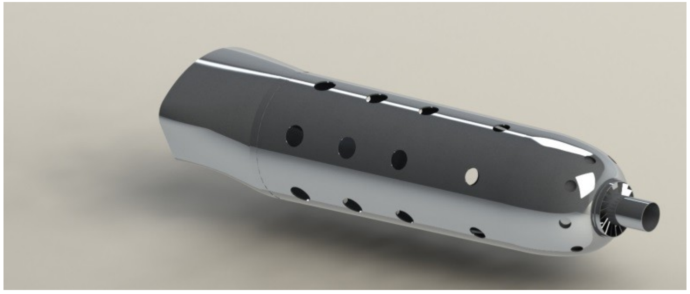

Figure 12: Combustor with Effusion Cooling

In this case multiple holes are employed in the combustor to cool the combusted gas.

Velocity Profile

- The velocity is minimum about 70m/s primary zone (figure 13) because of swirling effect. Once the combustion takes place, velocity increases along the length of chamber with approx. 500m/s at dilution zone and approx. 650m/s at outlet.
- High velocity (approx. 850 m/s) region is formed at the centre of nozzle guide vane.

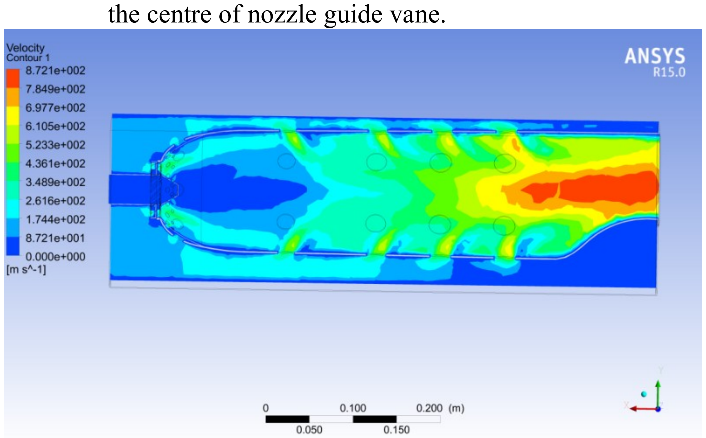

Figure 13: Velocity Contour

Temperature Profile

- Flame is observed to get stabilized well within the primary zone.
- Combustion temperature at Primary zone is about 1900K (figure 14), temperature increases towards the wall reaching maximum temperature of about 2000K near the wall of primary and intermediate zone (figure 20).
- Flame propagates axially along the length of flame tube with temperature approx.1900K (figure 15).
- Cooling holes in dilution zone consequently cools the flame temperature. The outlet temperature is approx. 1100K near the walls (figure 14), but small region at the centre with temperature approx. 1600K is formed.

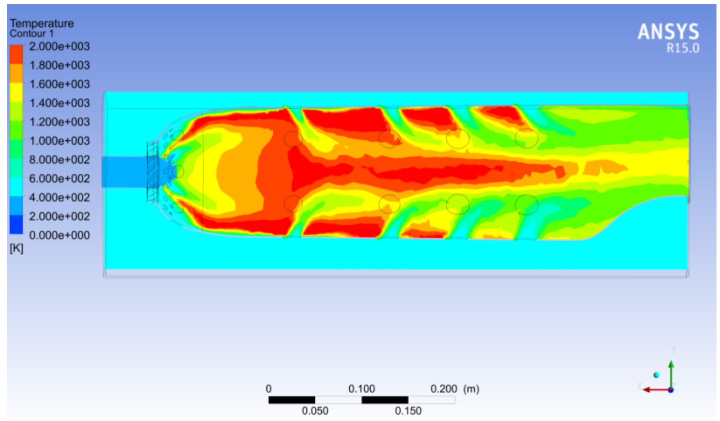

Figure 14: Temperature Contour XY Plane

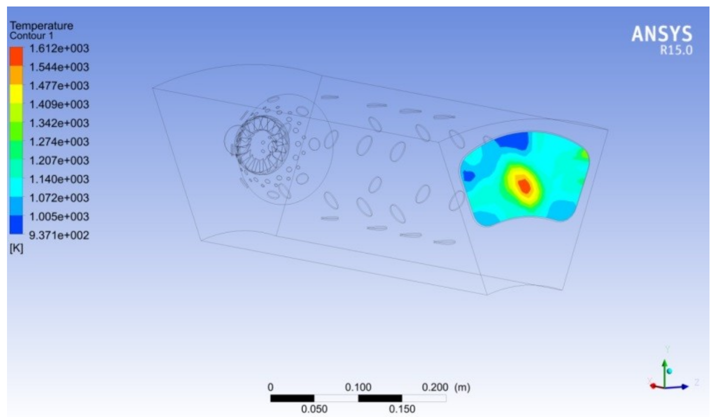

Figure 15: Outlet Temperature

Thermal Analysis

The obtained results from the flow analysis are imported to Ansys Thermal to check the surface temperature. The obtained results are shown below.

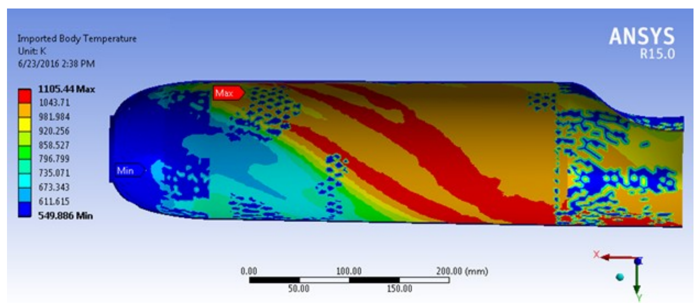

Figure 16: Surface Temperature of model without any cooling

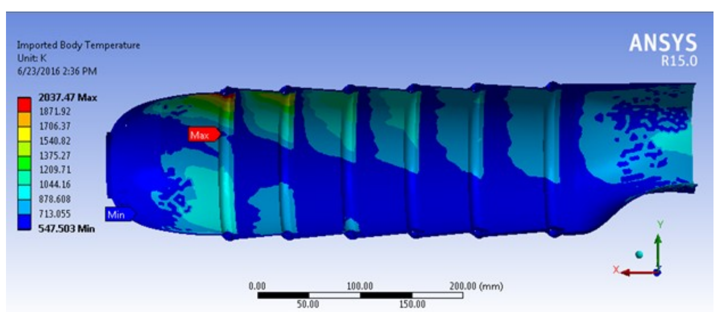

Figure 17: Surface Temperature of model with Film cooling

The above results are the surface temperature of different cooling process. Figure 16 shows the surface temperature of the model without any cooling, where we can notice that surface temperature is reaching to maximum 1105K which is optimum value but proper combustion is not taking due to lack of air. Surface temperature of combustor with Effusion cooling (figure 18) is above the melting point of the combustor material. Effusion cooling is provided to cool the temperature combusted gas. In figure 17 we can notice that the proper cooling ofsurface is taking place. Thus film cooling is essential to reduce the surface temperature of the combustor

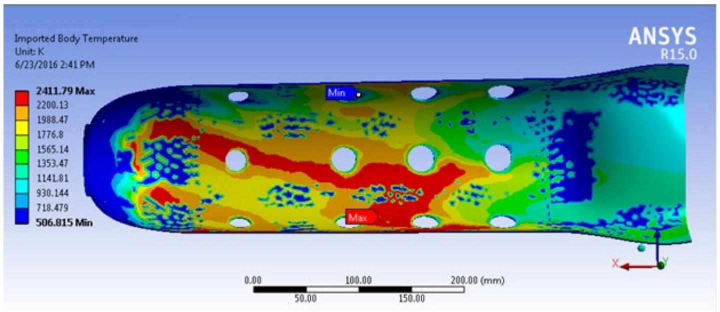

Figure 18: Surface Temperature of model with Effusion cooling

IV. CONCLUSIONS

Thermo-Fluid analysis of the combustion chamber is carried out using the CFD software Ansys. The effectiveness of different cooling methods(film cooling and effusion cooling), combustor model are presented below:

- **Combustion chamber with no cooling holes:** Complete and proper combustion didn't take place due to lack of primary air (through primary holes) to enhance combustion.
- **Combustion chamber with film cooling holes:** Combustion took place and the flame produced propagated along the centre (axis) of the combustor, i.e., no combustion took place near the chamber wall. It thus proved, film cooling enhances the surface cooling of combustion chamber.
- **Combustion chamber with effusion cooling holes:** Analysis was carried out with only effusion cooling holes (multi holed geometry) with no film cooling setup. The combustion flame propagated along the combustion chamber with regular cooling provided from the cooling holes present on the walls of chamber and flame temperature was gradually reduced. Unlike film cooling, the flame was spread towards the walls. This verifies, effusion cooling is for lowering the hot gas (flame) temperature.

To get stable fame and symmetric temperature profile distribution along the axis of combustor, it is intrinsic to have both film cooling and effusion cooling in the combustor geometry.

REFERENCES

[1] 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.

[2] 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.

[3] 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"

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

[5] Cooling of Combustion Chamber by MajaMunktell, Lund University, Project Report 2008 MVK160 Heat and Mass Transport, 2008.

[6] Leger B, Miron P and Emidio J.M, Geometric and areo-thermal influences on multi-holed plate temperature: application on combustor walls, Elsevier science Ltd, 2002.

[7] James J. Mcguirk, Jose Laginha Palma, 1993,"Flow inside model gas turbine combustor: Calculations", Journal of Engineering for Gas Turbines and Power - July 1993.

[8] H. I. Oguntade, G.E. Andrews, A. D. Burns, D. B. Ingham and M. Pourkashanian, "The Influence the Number of Holes on Effusion Cooling Effectiveness for an X/D of 4.7", Proceedings of ASME Turbo Expo 2015: Turbine Technical Conference and Exposition

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

[10] Philip G. Hill, Carl R. Peterson "Mechanics and Thermodynamics of Propulsion" Dorling Kindersley (India) Pvt. Ltd., pp. 250-255, 2010.

[11] Heat and mass transfer A practical approach (3^{rd} edition) YunusCengel, Published by McGrawHill, pp. 285-561, 2006.

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