International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 6, June - 2014

Reduction of Gas Turbine Combustor Pattern Factors using CFD

Ranjith P. V[1], M. Shivashankar[2],G. Sivaramakrishna[3], Vimala Narayanan[4]
[1]Project Trainee, GTRE, DRDO, Bangalore&M.Tech Student, SIT, Tumkur
[2]Associate Professor, Department of MechanicalEngineering, SIT, Tumkur
[3] Scientist 'F', [4] Scientist 'G' & Technical Director, GTRE, DRDO, Bangalore

Abstract:**The quality of hot gases emanating from gas turbine combustor is measured using two parameters viz., circumferential pattern factor and radial pattern factor. These parameters have a direct effect on the life of turbine vanes and blades and ideally they should be as small as possible. A higher total pressure loss in the combustor liner reduces the pattern factors but from the engine overall performance perspective, it has to be a minimum, as every one percentage increase in total pressure loss results in a half percent reduction in thrust and around a quarter of a percent increase in specific fuel consumption. Among all the variables that influence the pattern factors, the dilution hole area significantly alters the pattern factors. In the present work, effort has been made to reduce the pattern factors of a gas turbine combustor by altering the size of dilution hole without affecting the total pressure loss significantly. The commercially available CFD code 'Fluent' hasbeen used for the studies.**

Keywords:**Gas Turbine Combustor, Dilution Hole, Circumferential Pattern Factor, Radial Pattern Factor, CFD**

NOMENCLATURE

k	Turbulent kinetic energy, m^2/s^2
P_{inlet}	Mass averaged total pressure at combustor inlet, N/m^2
P_{exit}	Averaged total pressure at combustor exit, N/m^2
T_3	Averaged total temperature at inlet,K
T_4	Averaged total temperature at exit, K
$T_{max(\phi,r)}$	Maximum total temperature in the radial and circumferential directions at the combustor exit, K
$T_{avg\phi(r)}$	Circumferential averaged total temperature at a given radius at the combustor exit, K
ε	Rate of dissipation of turbulent kinetic energy, m^2/s^3

I. INTRODUCTION

Gas turbine combustor designs for aero applications are becoming increasingly challenging in order to meet the stringent requirements such as lower emissions, higher durability, lower maximum exit temperatures, lower fabrication and maintenance costs and reduced design and time-to-market cycle times. These requirements necessitate more emphasis on Computational Fluid Dynamics (CFD) simulation of the combustion flow field to reduce testing and improve performance, which, for practical aero-engine applications is a complex problem.

Improvements in engine performance come in the form of increasing thrust production while increasing the working life of the individual engine components. Increasing the thrust can be accomplished by increasing the gas working temperature of the turbine section. The fact that the combustor exit temperature, especially when it is non-uniform, has a drastic effect on the life of turbine blades, and hence the maintenance costs, makes it a critical design requirement. Temperature non-uniformity at the exit of the combustor is often referred to as hot streaks. The existence of hot streaks causes local hot spots on the blade surfaces, leads to heat fatigue of blade and reduces blade life.Pattern factor and profile factor of the combustor indicate the non-uniformity of the temperature at the exit of combustor which dictates the life of the turbine blades.A lower value of the same is always desirable.

Gas turbine combustor should possess a minimum value of total pressure loss as every one percentage increase in pressure loss can result in either a half percent reduction in thrust or around quarter of a percent increase in specific fuel consumption. It comprises of cold loss that include loss due to friction, turbulence and hot loss, which is due to heat addition. In a combustor, hot losses are unavoidable, whereas, cold losses can be reduced by suitably modifying the combustor geometry. However, some pressure loss is beneficial to the combustion and dilution processes, because it gives high injection air velocities and steep penetration and high levels of turbulence, which promotes good mixing and can result in a shorter liner. Equations 1.1, 1.2, 1.3 define the pressure loss,Circumferential Pattern Factor(CPF) and Radial Pattern Factor(RPF) respectively.

$$pressure\,loss = \frac{P_{inlet} - P_{exit}}{P_{inlet}} \tag{1.1}$$

$$CPF = \frac{T_{\max(\phi,r)} - T_4}{T_4 - T_3} \tag{1.2}$$

$$RPF = \frac{T_{avg\phi(r)} - T_4}{T_4 - T_3} \quad (1.3)$$

RPF and CPF indicate the non-uniformity of temperature at the combustor exit and it normalizes the difference between the maximum and mean exit temperatures [1].The quality of combustor exit temperature is influenced by many factors. Dilution and combustion chamber cooling air, the geometry of the combustion chamber, fuel spray characteristics and operating conditions all contribute to the pattern factor in varying degrees. These factors, expanded in Fig. 1, have a high level of interdependency, and it is therefore difficult to separate fully the results of each of their influences.

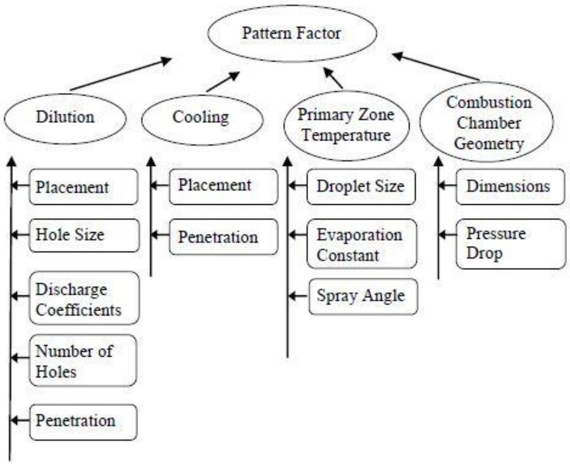

Figure 1. Contributors to pattern factor

According to Momtchiloff [2] the two parameters that strongly influence the combustion chamber outlet temperature profile are the combustion chamber length and the pressure loss across the combustion chamber. Sj¨oblom [3] highlighted theeffects of dilution air on pattern factor. He found that as the turbine inlet temperatureincreased, there was a reduction in the amount of air available for dilution purposes. This reduction in dilution air degraded the dilution zone mixing and resulted in an increase in pattern factor. That study showed that with a 15% increase in dilution air, the pattern factor was reduced from 0.35 to 0.2. This author also describes the three leading contributors to pattern factor. The first was the temperature profile which was created in the primary zone. The second was the behavior of the combustion chamber cooling air, which created peaks in pattern factor. Finally, the design of the dilution air holes.

George and Cox [4] performed analytical and experimental research into pattern factor. Using an annular combustion chamber,they investigated the effects of changing dilution and cooling air, dilution zone geometry and operating conditions. Theirconclusionsincluded the observation that pattern factor was mainly affected by the mixing processes within the combustion chamber and the temperature profile created in the primary zone.Clayton Kotzer et.al.,[5] have experimentally investigated the effect of combustor geometry on the exit temperature fields using an ambient pressure test rig. They concluded that relatively small geometric changes in dilution zone can lead to dramatic changes in the exit temperature field.

Lefebvre [6] carried out significant research into the parameters that effect pattern factor. He presented that pattern factor was mainly influenced by a combination of the number of dilution jets and their penetration depth into the combustion chamber. The overall combustion chamber geometry, pressure loss, discharge coefficient of all holes, and airflow distribution still played a role in the creation of the exit temperature profile, but to a lesser degree. He also commented that the temperature profile which enters into the dilution zone contributes to the pattern factor. This entering temperature profile is a function of the fuel spray characteristics of droplet size, evaporation constant and spray angle.

Kishorekumar et.al.[7], narratedthe application of CFD tools in the design and analysis of gas turbine combustors.Sivaramakrishnaet.al.[8], have carried out 3-D cold flow CFD analysis of an aero gas turbine combustor using the experimental data obtained through customized lab scale tests on an annular aero gas turbine combustor. Srinivasa Rao et. al.[9,10], have carriedout 3-D reacting flow analysis of an aero gas turbine combustor using 'Fluent'. They have also reduced the combustor total pressure loss through CFD analysis by modifying the shape of pre-diffuser struts [11].Motsamai et.al.,[12] used CFD and mathematical optimization to minimize the combustor exit temperature distribution.

1.1 Objective

In this work, through reacting CFD analyses, an effort has been made to reducethe pattern factors of a gas turbine combustor by altering the dilution hole size. During this process, it has been ensured that the total pressureremains almost the same. Analyses have been carried using Fluent, for three differentinner dilution hole diameters. Results obtained through computationswere compared with baseline combustor and the hole diameter that results in lower pattern factors has been suggested for improving the engine life.

II. COMPUTATIONAL MODEL

2.1 Combustor geometry

Combustor geometry taken for the analyses is an annular combustor having 18 atomisers and 18 swirlers equally spaced along the circumference. Reacting flow analyses have been carried out for the flow in a 20^0 sector combustor with an atomiser and a swirler as shown in Fig.2.

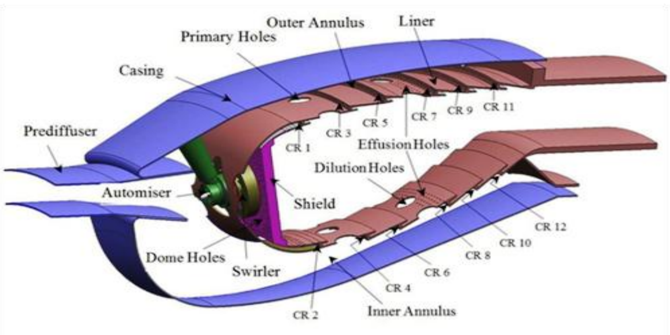

Figure 2.Sector view of computational domain

The baseline and various modified configurations ofinner dilution hole geometry that were considered for the analyses have been shown in figure 3
.

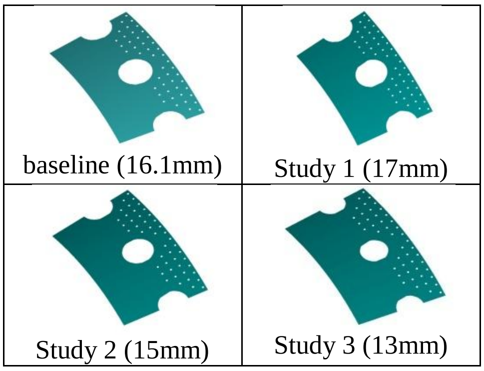

Figure 3.Configurations of inner dilution hole geometry analysed

2.2 Combustor Grid

A 3-d hybrid grid with a cell count of approximately 5.8 million has been generated for three different geometry using ICEM-CFD [13], the pre-processor of fluent. All the complex features viz., swirler vanes, liner holes, holes on dome, shield, effusion holes, cooling ring, and air blast atomiser air passages have been modelled exactly as per the geometry. Figure 4 shows 3-D view of overall grid generated.

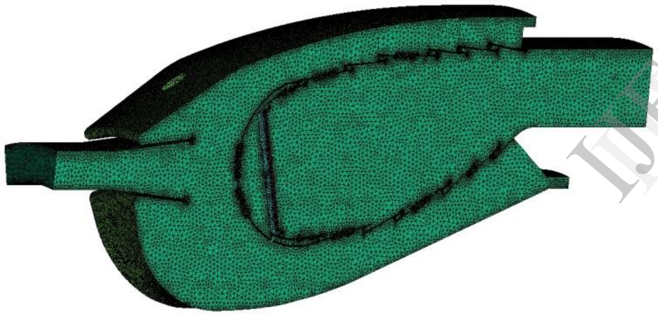

Figure 4.Overall grid generated

2.3 Grid Quality

While generating the grid, care has been taken to maintain the quality of mesh with regard to the aspect ratio, equi-angle skew-edge ratio and other parameters like equi-size skew and mid angle skew are also within acceptable range.

2.4Boundary condition

Total pressure and total temperature have been specified at the combustor inlet, static pressure along with the target mass flow rate has been specified at the combustor core exit. Turbulent intensity and hydraulic diameter have been specified as the initial conditions for the inlet turbulence. At the combustor bleeds where no combustion takes place, mass flow rate boundary condition has been specified in such a way that the prescribed quantity of flow goes out of the domain. All the combustor walls have been treated to be adiabatic. Periodic boundary conditions have been imposed on both the sides of the sector in the circumferential direction.

2.5. Governing Equations

In the present study, flow is treated to be steady, turbulent, compressible and reacting. The governing Navier-Stokes equations (RANS) for the conservation of mass, momentum, energy and species concentration for the gas, together with an equation of state are approximated for each mesh cell. The resulting sets of equations are solved numerically to obtain the flow field, mixing and combustion data.

2.6 Injection model

The aviation turbine fuel ($C_{12}H_{23}$) has been injected as discrete phase using cone injection model at the exit of the injector fuel passage. Fuel injection parameters used for analyses have been listed in table 1.

Table 1: Injection Parameters

Type of injection	Conical
Injection velocity	40m/s
Spray cone angle	80^0
Sauter mean diameter(SMD)	20 microns

2.7 Chemical Reaction Scheme

A key component of the reacting CFD analysis is a mathematical description of the combustion process and its interaction with the turbulent flow field. In general, numerical modeling of the combustion process requires detailed or reduced chemical reaction mechanisms for adequate accuracy. In the present study, Non premixed combustion model has been used for the simulation of chemical reaction for fuel-air mixture. $C_{12}H_{23}$ fuel chemistry is modeled by using a simplified two step chemical reaction scheme given below.

$$C_{12}H_{23} + 11.75\ O_2 \longrightarrow 12\ CO + 11.5\ H_2O \qquad (2)$$

$$CO + 0.5O_2 \longrightarrow CO_2 (3)$$

The reaction rate is calculated using a combined Arrhenius and Eddy Breakup model. The minimum of these two rates is taken into consideration.

2.8Turbulence model

Turbulence has been modeled using Realizable k-ε two equation model[13]. The values of turbulence intensity and hydraulic diameter have been specified as the turbulence initial conditions.

2.9 Numerical integration scheme

The partial differential equations for conservation of mass, momentum, energy, chemical species, turbulent kinetic energy and its dissipation rate are integrated over individual finite control volumes and the resulting volume integrals are transformed into their surface counterparts. The pressure-velocity coupling is achieved using SIMPLE (Semi Implicit Method for Pressure Linked Equations) algorithm.

III. VALIDATION OF THE CODE

Prior to this study, validation studies have been carried out using the experimental data obtained on the actual combustor hardware,a close agreement has been found between CFD and experiments in respect of total pressure loss, RPF and CPF [15].

IV. RESULTS AND DISCUSSION

The change in mass flow split on outer annulus, inner annulus and core region with respect to baseline (existing) combustor as a percentage of inlet mass flow rate, predicted by CFD has been compared for all the studies and shown in Table 5. The outer annulus mass flow is computed on a plane just before primary holes in the outer annulus and similarly the inner annulus mass flow is computed on a plane just before primary holes on the inner annulus. It is found that the mass flow rate on inner annulus region reduces as the dilution hole diameter is reduced.

TABLE 2.Change in % mass flow split with respect to baseline combustor

	% Change		
Study number	**Outer annulus**	**Inner annulus**	**Core**
Study 1	-0.3	+0.7	-0.4
Study 2	+0.6	-0.9	+0.4
Study 3	+1.4	-2.3	+0.9

The contour plot of non-dimensional total pressure for various studies on a plane in-line with the atomizer is shown in Figure 11, where a gradual decrease in total pressure along combustor length can be seen.

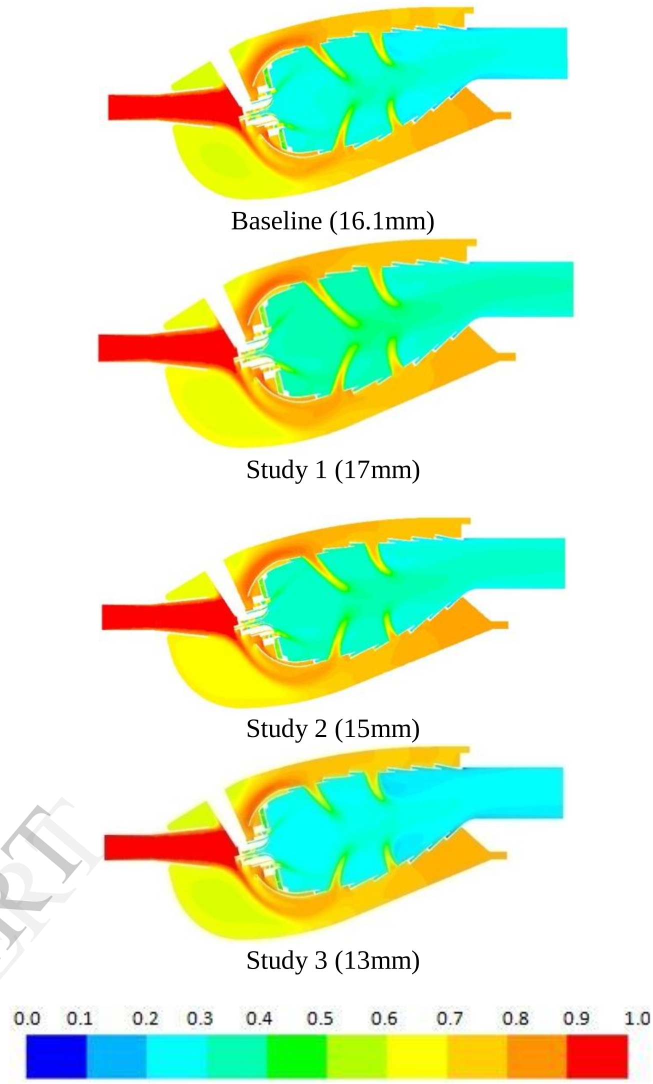

Figure 5. Comparison of total pressure contours on a plane in-line with atomizer

Change in % of total pressure loss with respect to baseline combustor for different studies is shown in table 6.These pressure loss values have been obtained at an inlet Mach number of 0.34. It is observed that overall pressure loss is maximum for study 3 and minimum for study 1. However, the change is marginal and insignificant.

TABLE 3.Change in % total pressure loss with respect to baseline combustor

		% Change		
Study number	**MN-Inlet**	**Outer annulus**	**Inner annulus**	**Overall**
Study 1		0.0	-0.2	-0.2
Study 2	0.34	-0.02	-0.2	+0.1
Study 3		0.0	-0.1	+0.2

Typical contours of non-dimensionaltotal temperature on a plane in-line with atomizer have been compared for Baseline, Study 1, Study 2and Study 3 configurationsand shown in figure 6. A local high temperature zone can be seen, downstream of the primary jets, from the below figure.

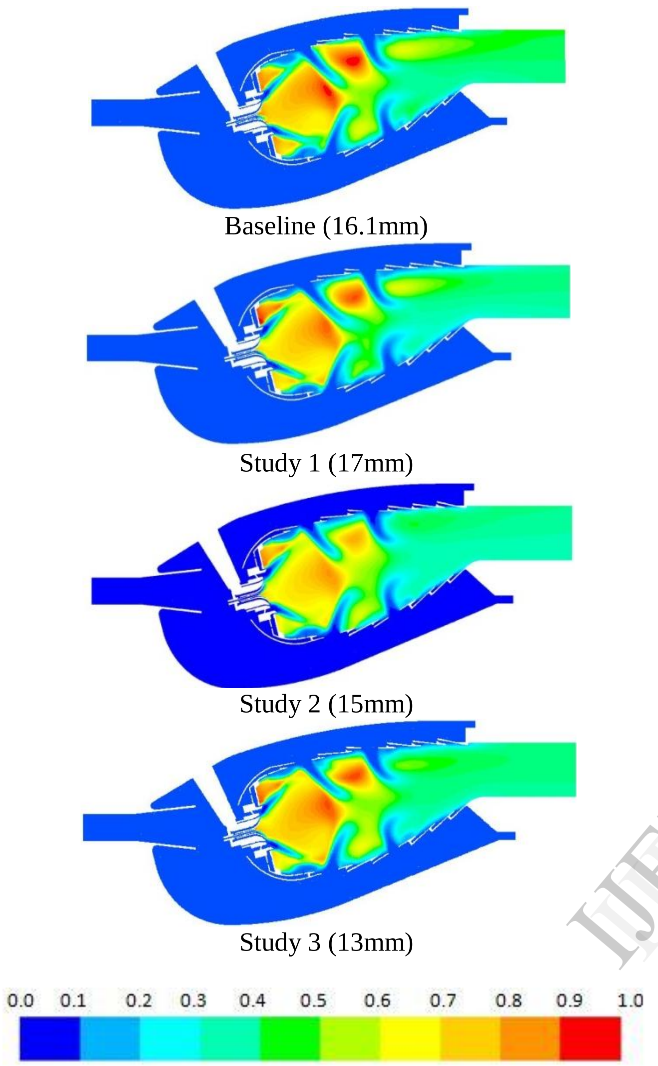

Figure6.Comparison of total temperature contours on a plane in-line with atomizer

Change in values of RPF, CPF, average & maximum total temperatures at combustor exit for all the combustor configurations have been listed in Table 6.

TABLE 4.Comparison of change intemperature distribution at exit plane with respect to baseline combustor.

Study number	Change			
	Max temp	**Mean temp**	**CPF**	**RPF**
Study 1	+37	+3	+0.05	0.00
Study 2	+20	+4	+0.02	-0.02
Study 3	-27	+3	-	-0.05

			0.04	

It is observed from the above table that a decrease in the dilution hole diameterreduces the local maximum temperature, RPF and CPF values at the exit plane.

Comparison of non-dimensionaltotal temperature on the combustor exit plane for baseline, study 1, study 2 and study 3 is shown in figure 9. From the below given figure, the high temperature region has been found in the vicinity of outer liner.

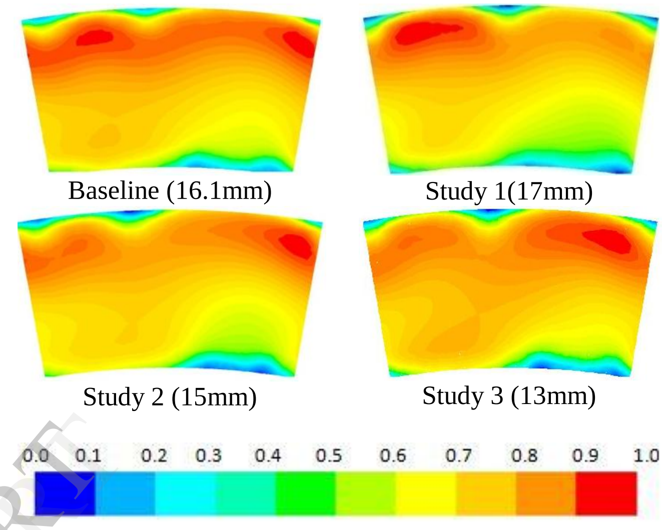

Figure.7Comparison of total temperature contours on the combustor exit plane

DISCUSSION

Study 1: with increase in dilution hole diameter to 17mm as shown in Table 2, the mass flow rate through the inner annulus has been increasedby 0.7% also reduction in the mass flow rate through the outer annulus and the core region has been observed. It can be seen from Table 3 that the total pressure loss decreases by about 0.2%.As shown in Table 4, there is no change in RPFvalues and CPF valuehas increasedby 0.05in comparison tothe baseline.

Study 2:With a reduction in the diameter of inner dilution holes to 15mm from 16.1 mm, as shown in Table 2, a reduction in the mass flow through the inner annulus has been observed and the same has been re-distributed in the outer annulus and core region. Table 3 shows an increase in the overallpressure lossby 0.1%. As shown in Table 4, RPF value reduced by 0.02and CPF value increasedby 0.02.

Study 3: With a reduction in the diameter of inner dilution holes to 13mm from 16.1 mm, as shown in Table 2, mass flow through the inner annulus has decreased and the same has been re-distributed in the outer annulus and core region. However, it can be seen from Table 3 that the total pressure loss increased by about0.2% and as shown in Table 4, RPF and CPF values have reduced by 0.05 and 0.04 respectively. These values show a significant improvement in the pattern factors and hence this modification of the combustor is a favorable configuration.

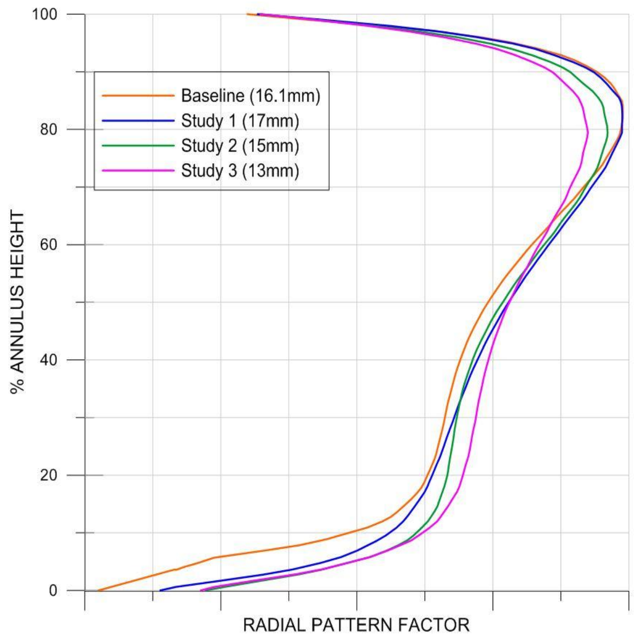

Figure 8. Comparison of the predicted RPF profile

The RPF curves along the annulus height for all three configurations have been plotted in figure 8. It can be seen from the figure that the peak of RPF profile occurs near82 % of the annulus height for all configurations and the value of maximum RPF reduces as the inner dilution hole diameter is reduced.

CONCLUSION

The inner dilution hole diameter has been varied to improve the pattern factors of an annular gas turbine combustor.

Among the studies carried out, a decrease in the inner dilution hole diameter has been found to be favorable for improving the pattern factors. A significant improvement in the CPF and RPF valueshas been seen byreducing the dilution hole size to 13mm. The resulting increase in overall total pressure loss due to this was found to be very less and insignificant. Hence the configuration with the diameter of inner dilution holes being 13mm can be suggested when enhancement of the engine life is the major criteria.

REFERENCES

[1] Lefebvre, A. H., 1998, "Gas Turbine Combustion" Second edition, Taylor & Francis.
[2] Momtchiloff, I., 1964, “The design and performance analysis of gas-turbine combustion chambers,” Report No. 1082-1, Volume.
[3] Sj¨oblom, B., 1980, “Some aspects on increasing gas turbine combustor exit temperature,” ASME Gas Turbine Conference and Products Show, pp. 10–13, 80-GT-73.
[4] George B. Cox, J., 1975, “Predicting exit temperature profile from gas turbine combustors,” Journal of Aircraft, 13, pp. 630–636.
[5] CalytonKotzer, Marc LaViolette and William Allan, 2009, "Effects of combustion chamber geometry upon exit temperature profiles", ASME Paper No GT2009 - 60156.
[6] Lefebvre, A. H., 1984, “Fuel effects on gas turbine combustion - liner temperature, pattern factor, and pollutant emissions,” AIAA/SAE/ASME 20th Joint Propulsion Conference, Cincinnati, Ohio, pp. 1–17, AIAA-84-1491.
[7] Kishorekumar, S., Venkataraman Shankar, Sivaramakrishna, G and Srinivasa Rao, M., “Application of CFD Tools in The Design and Analysis of a Gas Turbine Combustor”, 7th Asian CFD Conference, 2007.
[8] Sivaramakrishna, G., Muthuveerappan, N., Kishore Kumar, S., and Venkataraman Shankar., 2006, “3-D CFD Analysis of An Aero Gas Turbine Combustor and Validation With Lab Scale Experiments”, Proceedings of 8th National Conference on Air Breathing Engines and Aerospace Propulsion (NCABE – 2006).
[9] Srinivasa Rao, M., Muthuveerappan, N., Kishore Kumar, S., Venkataraman Shankar., 2006, “3D Reacting Flow Analysis of an Aero Gas Turbine Combustor using Fluent”, Proceedings of 2006 Fluent Users Group Meeting.
[10] Srinivasa Rao, M., Sivaramakrishna, G., Kishore Kumar, S., Venkataraman Shankar., 2007, “CFD Analysis of an Aero Gas Turbine Combustor – Validation and Experiments”, Proceedings of ICONICE 2007 Conference.
[11] Srinivasa Rao, M., Sivaramakrishna, G., Kishore Kumar, S., Venkataraman Shankar., 2007, “Reduction of Combustor Total Pressure Loss using ‘Fluent’ by modifying the shape of Pre-Diffuser Struts”, Proceedings of 2007 Ansys India Conference.
[12] Motsamai, O.S., Visser, J.A., Morris, M., DeKock, D.J., 2006, “An Efficient Strategy for the Design Optimization of Combustor Exit Temperature Profile”, ASME Paper No. GT2006 – 91325.
[13] ICEM User’s Guide. Version 14.5.7
[14] T.H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu, 1995, “A New k-e Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation”, Computers Fluids, 24(3), pp. 227-238.
[15] Srinivasa Rao, M., Sivaramakrishna, G., 2009, "Performance Improvement of a an aero Gas Turbine ", ASME Paper No GT 2009-59928.