



International Journal of ChemTech Research CODEN( USA): IJCRGG ISSN : 0974-4290 Vol.5, No.2, pp 912-917, April-June 2013

## ICGSEE-2013[14<sup>th</sup> – 16<sup>th</sup> March 2013] International Conference on Global Scenario in Environment and Energy

# Reacting Flow And Emission Characteristics In A Liquid Fuelled Aircraft Engine Combustor

## Pandey V.<sup>1</sup>\*, De A.<sup>1</sup> and Kushari A.<sup>1</sup>

<sup>1</sup>Department of Aerospace Engineering, Indian Institute of Technology, Kanpur, U.P., India-208016

> \*<sup>1</sup>Corres. Author: vivekp@iitk.ac.in Phone Number: +91-0512-259-7126

**Abstract:** In the present study, reacting flow and emission characteristics in a can combustor is numerically simulated and the results are compared with the data available in open literature. For reacting flow, continuity, momentum and transport equations for mean mixture fraction (f) and mixture fraction variance (f'') are solved. The equations for mean mixture fraction and mixture fraction variancelink the instantaneous thermochemical state of the fluid through this conserved scalar and its variance. For reacting flow simulations, the flow patterns for different drop size distributions, effect of different domain length, fuel variation and initial high inlet air temperature as a model for ignition have been studied. Aircraft emissions are becoming a critical constraint for combustion chamber design because of prevailing stringent emission norms and a first order estimate of emissions has been obtained in terms of temperature profiles. Emissions are closely related to fuel spray drop size distribution via temperature and combustor residence time. Hence study of drop size distribution and the related temperature pattern is essential for gaining insight into emissions formation process. **Keywords:** Emissions, Gas Turbine, Combustion, Liquid Fuel.

## Introduction

The gas turbine combustion system design and development cycle is an exercise to get an optimum solution for the conflicting design requirements of gaseous emissions and combustion efficiency .Space and weight are primary constraints in aircraft engine design and much effort is made to reduce combustion chamber volume by using energy denseliquid fuels that have high specific energy (of the order of 45 MJ/kg). In addition, storage and transport of liquid fuelsare safer and easier compared to gaseous fuels<sup>1</sup>. Hence liquid fuels are indispensable for aircraft applications. Moreover, combustion intensity is to be maximized in order to keep the engine size small. The heat release rate in aero-gas-turbines is of the order of  $5 \times 10^8$  Watt/m<sup>3</sup>, which is about 100 times as compared to a large stationary power plant furnace. One reason for the difference is that the density of reactants in aircraft gas turbine is about ten times higher than in an atmospheric-pressure furnace; another reason is the fineness of atomization of the injected fuel and also the turbulence intensity. Additionally, stringent emission requirement restrict the lean and rich limits for fuel air ratio. Fuel rich condition is needed for high power but it would create higher unburnt hydrocarbons (UHC) whereas fuel lean burning will decrease NOx but CO, UHC

will increase when compared with stoichiometriccondition or optimum temperature range. Lean burn also carries with it additional risk of thermo-acoustic combustion instabilities. Above factors make the combustor design for civil or military aircraftsa challenging task.

Design process for majority of gas turbine combustors is based on empirical correlations derived from available experimental data. Since the experiments have inherent limitations, e.g. time consuming, expensive, limited data, etc., Computational Fluid Dynamics (CFD) simulations have been found to be a good source of data for complementing experiments and reducing design cycle time. Recent advances in the capability of CFD codes have led to their application<sup>3,4</sup> in complex three dimensional flows in gas turbine combustors.

An aero-engine combustor can be viewed as consisting of two parts-an outer annular supply casing and a cylindricalcombustion chamber liner. Primary, secondary and dilution holes in the liner connect the annular space to the combustion chamber.Traditionally<sup>5</sup>,CFD predictions for these two partshave been done separately using a 'weakly coupled' approach.The flow conditions at the liner holes are derived from one-dimensional correlations which are used as boundary conditions for a combustor flow fieldprediction. The above practice may be deficient in at least two important aspects<sup>6, 7</sup>. Firstly, the solutions are strongly dependent on the accuracy of boundary conditions. Secondly, no information is provided on velocity profiles or turbulence conditions at the liner hole entry locations. Therefore, the above practice is not recommended for accurate design. For example, the effects of variation in inlet conditions on the properties of the flow entering the liner holes and thereby altering the primary zone flow pattern inside the combustion chamber are well observed<sup>8</sup>.To reduce the above uncertainty, modelling of the entire flow field from compressor exit to turbine inlet has been carried out by some researchers<sup>9-11</sup>.

Some researchers have resorted to atmospheric pressure simulations for gas turbines. This is not relevant for in-flight aircraft engine operation. A reacting flow calculation with in-flight conditions is required for practically useful characteristics such as stability loop and combustion efficiency. Therefore, the primary aim of the present study is to obtain flow, temperature and emission characteristics in a can type combustor without de-coupling the outer casing and the can flow, by using three-dimensional geometry and actual in-flight conditions. Pressure drops across the combustor have been reported in a previous study<sup>12</sup>along with radial velocity, axial velocity, temperature and turbulent kinetic energy profiles at different cross-sections. A high altitude flight condition (Reynolds Number = 122000, at the combustor inlet and Fuel Air Ratio = 0.011) is simulated.

#### **Numerical Details**

Three dimensional, steady and unsteady, turbulent reacting flows are simulated using ANSYS-Fluent-13<sup>13</sup>.Grid Cross-Section is shown in Figure-1 The details of computational domain, selected physical models, boundary conditions and solution methods are described in the following sub-sections.

#### **Computational Domain**

For the purpose of investigating the flow field and combustion characteristics inside a can combustor, a threedimensional model of a single 'can' of a can-annular aircraft engine combustor<sup>14</sup> has been modelled. An axial cross-section (Fig. 1) shows the relative locations of primary holes, louver holes, dilution holes and swirler. Grid-Independence test along with mass split distributions and turbulence model has been done and presented previously<sup>25</sup>.

#### **Turbulence-Chemistry Interaction**

The reacting flow calculations are done with kerosene ( $C_{12}H_{23}$ ). A non-premixed combustion model is applied for droplet combustion with kerosene. In the non-premixed modelling approach, the instantaneous thermo-chemical state of the fluid is related to a conserved scalar quantity known as the mixture fraction, f, which can be written in terms of the atomic mass fraction<sup>15</sup>. Species transport equations can be reduced to a single equation in funder the assumption of equal thermal and molecular diffusivities. Conservation equations for mean mixture fraction ( $f_{mean}$ ) and mixture fraction variance (f'') are solved<sup>16,17</sup>. The mixture fraction variance is used in the closure model for turbulence-chemistry interactions. A -pdf (probability density function)<sup>18-20</sup> method is applied for the closure as it very closely represents experimentally obtained pdf's for a wide range of conditions<sup>21</sup>.

## **Boundary Conditions**

A mean mixture fraction equal to zero for oxidizer and one for  $C_{12}H_{23}$  is applied at the fuel boundary which is composed of points at which the droplets start to evaporate. The discrete phase trajectory for evaporating droplets is found by a Discrete Particle Model (DPM) using a Lagrangian formulation. Mass, momentum and energy exchanges for the droplets are calculated by integrating across control volumes while the interaction of the droplets with the continuous phase is also taken into account.

The droplet spray conditions used in the present study are as follows: droplet velocity = 6 m/s; half spray cone angle equal to  $73^{\circ}$ ; a Logarithmic Rosin-Rammler drop size distribution with maximum, minimum and mean-droplet diameter's equal to 80, 5 and 40 microns respectively<sup>22</sup>. Atomizer is modelled as a hollow cone with a diameter of 200 microns. Turbulent dispersion of particles is modelled with a discrete random walk model.

#### **Results and Discussion**

The effect of initial droplet distributions provided at the atomizer is shown in Figure 2. The results are reported for section z=1.22L. Details for drop distributions for Figure 2 are tabulated and reported in Table-1. The case c13-r-01 has lowest pattern factor for  $C_{12}H_{23}$  with a spread parameter of 3.5. Lowering the spread parameter (c13-04-a-rerun) or making the droplet size distribution uniform has an effect of increase in peak temperature and pattern factor. Diesel ( $C_{10}H_{22}$ ) is used as liquid fuel in the case c13-03-a (Table 1) where a higher pattern factor is noted in comparison with cases using  $C_{12}H_{23}$  as the fuel. Thiscan be because  $C_{10}H_{22}$  has higher boiling point than  $C_{12}H_{23}$ . This would mean that liquid fuel droplets stay for a longer part of combustor length before complete vaporization. Therefore, more evaporated fuel is available at downstream positions where some part of combustion occurs. This means a larger peak temperature profile.Higher Pattern Factor was also noted with droplet diameter greater than 40 microns. This is expected as droplet evaporation time increases in direct proportion to the square of droplet size.

In Figure 3, the effect of initial higher inlet-air temperature is shown. At the air inlet boundary condition, a higher temperature is set for one flow time (1000 iterations). Temperature is set to its actual (lower) value after this. This has the same effect as an ignition source which supplies energy for evaporation of fuel droplets as well as for the reaction kinetics occurringduring and after ignition. This provides improved simulation results when chemical kinetics for multistep chemistry istaken into account. Low peak temperatures at the exit are observed with highair inlet temperature for initial part of the simulation.

In Figure 4, temperature distribution is shown at different locations between exit section (z=0) and downstream of exit (z=1.44L/12a). Turbulent mixing results in a flatter profile at the downstream sections where pattern factors are calculated.

In Figure 5, effect of increasing the length of simulation domain from z=L (case a11) to z=1.44L (case a22) on the exit (location z=L)temperatureprofile is noted. There is an increase of peak temperature and also increase of average temperature for a22 (domain with larger length=1.44L) as compared with a11 (domain with a length of z=L). Pattern factor is 0.8 % larger with extension (a22) than without it (a11).Simulation domain extension may have considerable impact on flow characteristics. Flow will be restricted in axial direction near the exit region since constant pressure boundary condition is applied there. This is due to zero axial pressure gradient (dp/dr=0) which can be derived from above boundary condition. Therefore vortices or swirl are not captured near exit. This implies that turbulent mixing might beunder predicted. It is possible that convective boundary condition at exit may impact the flow field substantially and for this convective boundary condition is to be tested with different length domains for non-reacting flow.Rest of the simulations were done using extended length domain are the ones that enter the guide vanes. The temperature profile at inlet section of guide vanes is of particular interest as the nozzle guide vane design depends on the maximum temperature attained at vane inlet section. There are stationary and moving vanes (blades) in turbines. The material for these vanes must be able to withstand maximum temperature attained at the vane inlet cross-section.



The pattern factor may be defined as:

$$PatternFactor = \frac{T_{\max imum, exit} - T_{mean, exit}}{T_{mean, exit} - T_{inlet}}$$

Exit temperature profile has been studied in terms of pattern factor and it has direct impact on emissions. NOx formation is increased at higher temperature and higher fuel residence times because it is a slowly occurring reaction. Conversely CO and UHC formation will increase at lower temperatures. Therefore one can obtain an estimate of emissions by having only such elementary data as temperature profile.

Case	Maximum drop diameter (microns)	Mean Drop Diameter (microns)	Minimum drop diameter (microns)	Spread Parameter	Empirical Formula of Fuel	Pattern Factor
c13-02-a-	40	40	40	-	$C_{12}H_{23}$	0.817
rerun-2					_	
c13-04-a-	40	20	5	2.0	$C_{12}H_{23}$	0.589
rerun						
с13-03-а	40	20	5	3.5	$C_{10}H_{22}$	0.705
c13-r-01	40	20	5	3.5	$C_{12}H_{23}$	0.514

Table-1 Droplet Distributions and their effect on pattern factor.

### Conclusions

Reacting Flow simulations inside an aero-engine can-combustor have been carried out from the inlet to combustor outlet. The effect of spray distributions, fuels and extended computational domain on radial temperature profiles has been noted on temperature and emissions. Optimum droplet distributions exist for efficient combustion. Lower spread parameter or uniform fuel drop size distribution uniform has an effect of increase in peak temperature and pattern factor. Therefore one can expect higher NOx emissions in such cases.

Extending the computational domain has a slight effect on turbulent mixing but changing the exit boundary condition to convective type in place of constant pressure may have substantial impact on flow field and this can be noted more clearly with extended length of domain.

At the exit, low peak temperatures are noted when high air inlet temperature for initial part of the simulation is used. This is one of several numerical methods by which ignition can be modelled in reacting flows. A more realistic prediction of temperaturehas beenobtained using ignition source.

Temperature profiles obtained from various test cases are a direct indicator of emissions. NOx, CO and UHC formation are strong functions of temperature profile and combustor residence time or reaction time.

Aircraft emissions are becoming a critical constraint for combustion chamber design because of prevailing stringent emission norms and a first order estimate of emissions is achieved in terms of temperature.Emissions are closely related to fuel spray drop size distribution via temperature and combustor residence time.Hence study of drop size distribution and the related temperature patternis essential for gaining insight into emissions formation process.

## Acknowledgement

The funding and support extended towards this project by DST (Department of Science and Technology) and PWC (Pratt & Whitney Canada) is acknowledged.

### References

- 1. Weinberg, F., 1974, "The first half million years of combustion research and today's burning problems," Plenary Lecture, 15th (Int.) Symposium on Combustion, Tokyo, Japan.
- 2. Holdeman, J. D., Mongia, H. C., and Mularz, E. J., 1988, "Assessment, Development and Application of Combustor Aerothermal Models", NASA-TM-100290.
- 3. Motsamai, O. S., Visser, J. A., Morris, M., De Kock, D. J., 2006, "An Efficient Strategy for the Design Optimization of Combustor Exit Temperature Profile," GT2006-91325, ASME Turbo Expo 2006: Power for Land Sea and Air, May 8-11, 2006, Barcelona, Spain.
- 4. Liu, N. S., Shih, Wey, T. H., "Comprehensive Combustion Modeling and Simulation," Recent Progress at NASA Glenn Research Center, ISABE-2007-1268
- 5. Grobman, J. S., Diltrich, C. C., and Graves, C. C., 1957, "Pressure Drop and Air Flow Distribution in Gas Turbine Combustors," Trans. ASME 79, pp. 1601-9.
- 6. McGuirk, J. J., and Spencer, A., 1993, "CFD Modeling Of Annulus/Port Flows," ASME Paper 93-GT-185.
- 7. McGuirk, J. J., and Spencer, A., 1995, "Computational Methods For Modelling Port Flows In Gas-Turbine Combustors," ASME Paper 95-GT-414.

- 8. McGuirk, J. J., and Spencer, A., 2000, "Coupled and Uncoupled CFD Prediction of the Characteristics of Jets from Combustor Air Admission Ports," ASME Paper 2000-GT-125.
- 9. Lai, M. K., Reynolds, R. S., and Armstrong, J., 2002, CFD-Based, Parametric, Design Tool for Gas Turbine Combustors from Compressor Exit to Turbine Inlet, ASME Paper GT-2002-30090.
- 10. Eccles, N. C., Priddin, C. H., 1999, "Accelerated Combustion Design Using CFD," XIV ISABE Paper 99-7094.
- Crocker, D. D., Nickolaus, D., and Smith, C. E., 1999, "CFD Modeling of a Gas Turbine Combustor from Compressor Exit to Turbine Inlet," ASME-98-GT-184, Jrnl. of Engg. for Gas Turbines and Power, 121, pp. 89-95.
- 12. Pandey, V., and Kushari, A., "Aerothermal Design of Aero-Engine Can Combustor using Computational Fluid Dynamics," 2011, Twenty Fifth National Convention of Aerospace Engineers, NCAE2011, November 4-5, 2011, BIT Mesra, Ranchi.
- 13. Ansys Fluent 13.0 User's Guide, 2010, Ansys Inc., Canonsburg, PA, USA.
- 14. Sampath, P., Shum, F., 1985, "Combustion Performance of Hydrogen in a small Gas Turbine Combustor," Int. J. Hydrogen Energy, 10:12, pp. 829-837.
- 15. Sivathanu, Y. R., Faeth, G. M., 1990, "Generalized State Relationships for Scalar Properties in Non-Premixed Hydrocarbon/Air Flames," Combustion and Flame, 82:211-230.
- 16. Jones, W. P., and Whitelaw, J. H., 1982, "Calculation Methods for Reacting Turbulent flows: A Review," Combustion and Flame, 48, pp. 1-26.
- 17. Bilger, R. W., 1980, "Perturbation analysis of turbulent non-premixed combustion," Combust. Sci. Technol., 22, pp. 251–261.
- 18. Pope, S. B., 1990, "Computations of Turbulent Combustion: Progress and Challenges," Twenty-Third Symposium (International) on Combustion, The Combustion Institute, pp. 591-612
- 19. Pope, S. B., 1985, "Pdf methods for turbulent reactive flows," Progress in Energy and Combustion Science," 11(2), pp. 119-192.
- 20. Warnatz, J., Mass, U., Dibble, R. W., Combustion, 4th edition. Springer-Verlag.
- 21. Libby, A., Williams, F. A., 1994, "Experimental Investigation of a Premixed Flame in an Impinging Turbulent Stream," Twenty-Fifth Symposium (International) on Combustion, The Combustion Institute, pp. 1207-1214.
- 22. Lefebvre, A. H., "Gas Turbine Combustion," 1999, 2nd edition, Taylor and Francis.
- 23. Hill, P. G., Peterson, C. R., "Mechanics and Thermodynamics of Propulsion," 2<sup>nd</sup> edition, Addison-Wesley.
- 24. Pandey, V., De, A., Kushari, A., "Numerical Investigation Of Combustion Characteristics In A Liquid Fuelled Can Combustor," Proceedings of ASME 2012 Gas Turbine India Conference, GTIndia2012, Mumbai.

\*\*\*\*