© OCT 2018 | IRE Journals | Volume 2 Issue 4 | ISSN: 2456-8880

Preliminary Design and Numerical Simulation of a Reverse Flow Annular Combustor at Basic Design Point Operating Conditions

DEEPA M S [1], DR. A AROKKIASWAMY[2]

[1]*Research Scholar and Associate Professor, Department of Aeronautical Engineering, S J C Institute of Technology, Chikkaballapur*

[2]*Professor and Head, Department of Aeronautical Engineering, Dayananda Sagar College of Engineering, Bengaluru*

Abstract -- This paper presents the preliminary design methodology for a Reverse flow annular combustor based on calculation of the geometric parameters, gas temperature profile, liner wall temperatures and position of air admission holes for a small gas turbine engine with Jet A as fuel. It also includes the modelling and numerical simulation of temperature distribution at centerline, inlet and outlet of the combustor using Ansys-CFX, carried out at basic design point operating conditions.

Indexed Terms: Reverse flow annular combustor, numerical simulation, Ansys-CFX, operating conditions

I. INTRODUCTION

Reverse flow combustor is one in which, the flow leaves the combustion chamber in the opposite direction to the conventional combustion chamber. It reduces the length of the engine and is used mainly in engines whose last compressor stage is centrifugal flow type. Reduced length will allow single shaft sitting on two bearings instead of three, which reduces the vibrations and maintenance problems. The reverse flow process also allows warmer air to serve as the dilution air to control the NOx formation instead using other energy to preheat the dilution air or use cold air which could quench the flame and produce CO.

The operating parameters for a combustor is based on the basic geometric parameters of combustor, gas temperature profile, liner wall temperatures and position of air admission holes in the three zones of combustor.

The aim of this work is to design a Reverse flow annular combustor based on calculation of the geometric parameters, gas temperature profile, liner wall temperatures and position of air admission holes for a small gas turbine engine. It also includes the modelling and numerical simulation of temperature distribution at centerline, inlet and outlet of the combustor using Ansys-CFX, which is carried out at basic design point operating conditions.

II. DESIGN PROCEDURE

The preliminary combustor design procedure proposed in this work is based on the Melconian and Modak (1985)[1] model to calculate the gas temperature inside the combustor. The methodology assumes that the inlet combustor conditions are known from the engine cycle analyses.

The basic geometric parameters of combustors as: the total length of the combustor, length of each zone of the combustor, diameter or height of the flame tube and the casing, size of primary and secondary air admission holes, temperature profile is calculated using this procedure. The design methodology for a gas turbine combustor operating with different types of fuels is presented Fig. 1.

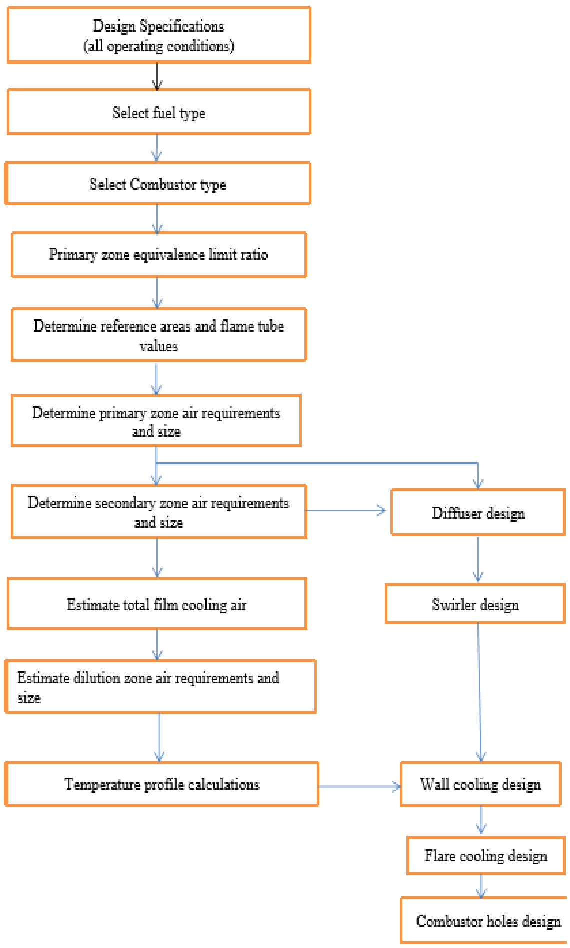

Figure 1: Proposed Preliminary design procedure

(Melconian and Modal 1985)

III. BASIC DESIGN OF REVERSE FLOW COMBUSTOR

The design of the combustion chamber was carried out using the initial conditions as obtained from the compressor outlet. The combustor design parameters are listed as from Brayton cycle analysis.

Table below summarizes the main characteristics of a small gas turbine engine.

Table I: Characteristics of a small gas turbine engine

Parameter	Value
Speed	51300 rpm
Compressor isentropic efficiency	77%
Compressor outlet temperature, T3	435 K
Compressor outlet pressure, P3	321 kPa
Turbine inlet temperature, T4	1135 K
Fuel	Jet A

The dimensions of a combustor might be determined either by aerodynamic or by chemical rate control. Generally, when the combustor is sized for a specific pressure loss, it will be sufficient to accommodate the chemical process too. However, it is necessary to verify all possibilities before the final choice. Thus, it is recommended that attention is given about the aerodynamic and chemical considerations.

The basic configurations of the reverse flow combustor were obtained based on the preliminary design procedure and the gas turbine characteristics. The configurations thus determined is as shown in table.

Table II: Basic configurations of Reverse Flow Combustor

Parameter	Results
Reference diameter, D_{ref}	0.263 m
Flame tude diameter, D_{ft}	0.250 m
Internal diameter, di	0.152 m
Length of the dome	0.0213 m
Swirler diameter	11.5 mm

Combustor Zone Length: The primary zone length (Lpz) shall be within 2/3 or ¾ of Dft. Similarly, a value of a half of Dft shall be taken for secondary zone length (Lsz). Total length of the flame tube shall consider the traverse quality(TQ) of temperature distribution on the combustor exit in order to preserve the turbine vanes. A method of calculation of total length is given in Lefebvre (1983) and a TQ of about 15% was considered. Dilution zone is the difference between total length and the sum of the primary and secondary zone length.

Table III shows the total length of the flame tube, primary, secondary and dilution zones of the designed reverse flow combustor.

Table III. Flame Tube zone length

Total Length (m)	Primary zone length (L_{PZ}, m)	Secondary zone length (L_{SZ}, m)	Dilution zone length (L_{DZ},m)
0.173	0.0583	0.050	0.065

An iterative process after determining the air mass flow rate that enters into each zone is used to determine the size and number of admission holes. This iterative process occurs as the discharge coefficient is unknown. Using the sequence of calculations described in Lefebvre and Mellor the number and diameters of the admission holes is determined as shown in table IV.

Table IV: Admission Holes

Parameter	Primary zone	Secondary zone	Dilution zone	Cooling
No. of holes	40	80	12	12
Diameter (mm)	8	6	24	12

IV. DESIGN METHODOLOGY

Based on the above obtained dimensions a reverse flow annular combustor was designed and the model is as shown in figure 2 which is further simulated using Ansys CFX

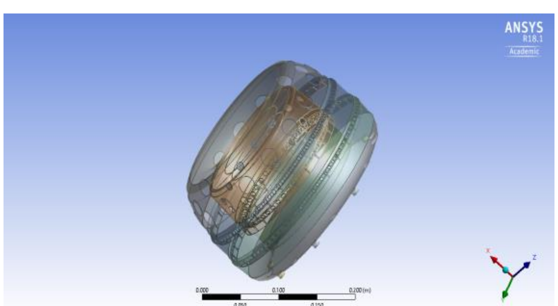

Figure 2: Model of the Reverse Flow Annular Combustor

The above model with 16 fuel injectors is cut into a section with one burner is considered for further simulation and is as shown in figure 3.

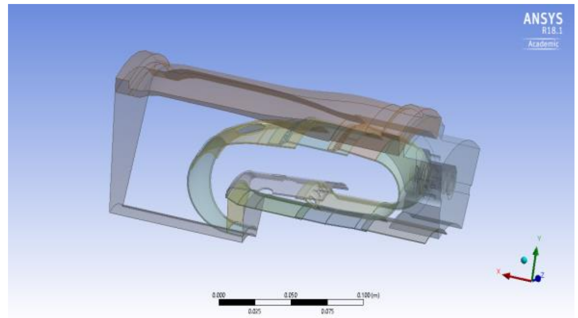

Figure 3: Section of the combustor for simulation

Ansys-CFX a commercial tool is used for the numerical simulation, the closed inspection of which suggest that many aerodynamic features are common to all systems. The main objective is to reduce the flow velocity and distribute air in prescribed amounts to all combustion zones.

The focus is done on the attainment of large scale flow recirculation for flow stabilization, effective dilution of combustion products and efficient use of cooling air along the liner walls. Visualization of flow under any conditions is made possible for successful operation of a combustor. due to recent advancement in computational fluid dynamics

The grid generation has been done in CFX-Mesh for this investigation. The mesh is composed primarily of tetrahedral mesh elements and the number of elements are about 3,50,000 with the scaling factor of 1:1:1. Figure shows the generated mesh of the combustor using CFX-pre.

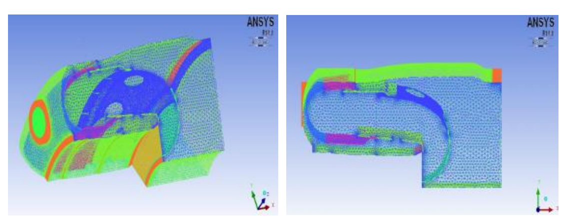

Figure 4: Meshed model of the combustor section

Boundary conditions are applied at the inlet of the combustor and the fuel injector, the flow is allowed to divide itself into liner and casing and from the casing into different zones through air admissions holes and cooling slots, which is a replica of the real case experimentation. The boundary conditions considered as shown in Table below.

Table V. Inlet Boundary Conditions

Sl. No.	Parameter	Unit	Values
1.	Air mass flow rate	Kg/s	0.800
2.	Fuel	Kg/s	0.01551
3.	Air Fuel Ratio	---	51.58
4.	Total pressure	kPa	320.998
5	Total Temperature	K	434
6.	Density	Kg/m^3	2.80
7.	Velocity	m/s	60

Shear Stress Transport Turbulence model is considered with Eddy Dissipated Transport equation for combustion and the combustor with boundary conditions is as shown in figure 5.

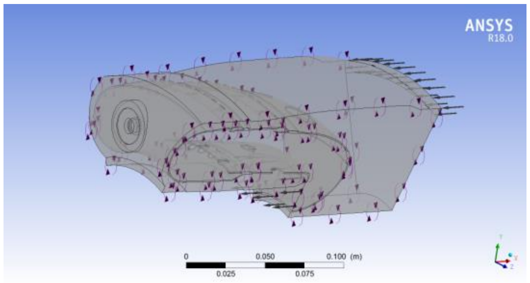

Figure 5: Combustor model with boundary conditions

V. RESULTS AND DISCUSSIONS

The below figures show the results obtained from the numerical simulation in CFX-Post. They show the total temperature distribution, density, Eddy Viscosity and Velocity profile in the entire region pf the combustor and at the combustor outlet. This phenomenon can be explained on the basis of the fact that the fuel mixing and evaporation of fuel takes place thereby indicating reduction in temperature levels. While an evaporation gets completed and more and more air is available from air admission holes through the performance of the combustor is improved and maximum expected levels are reached.

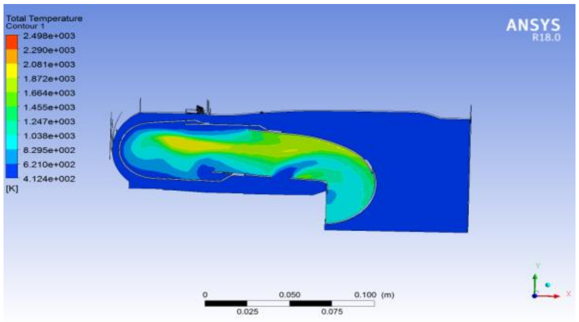

Figure 5: Total Temperature distribution throughout the length of the combustor

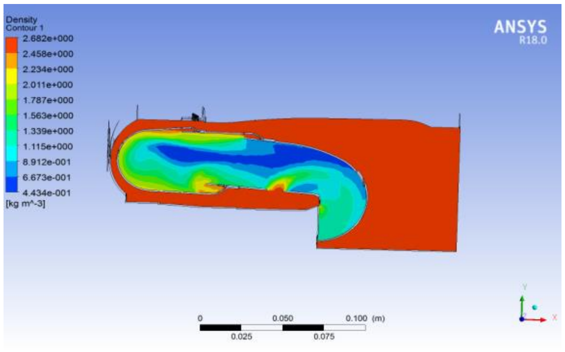

Figure 6: Density Profile throughout the length of the combustor

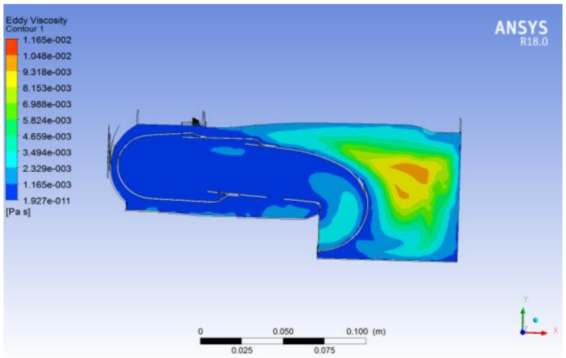

Figure 7: Eddy Viscosity profile throughout the length of the combustor

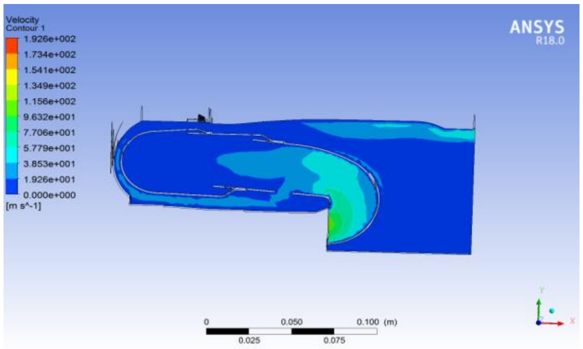

Figure 8: Velocity profile throughout the length of the combustor

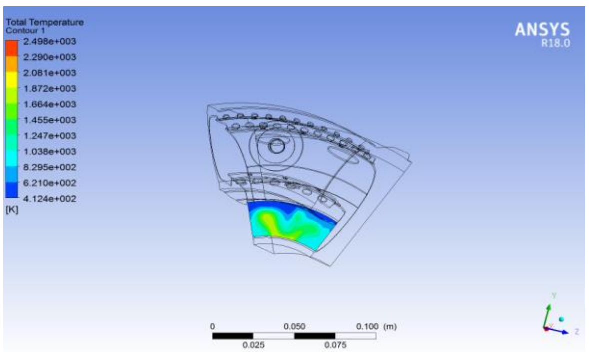

Figure 9: Total Temperature distribution at the outlet of the combustor

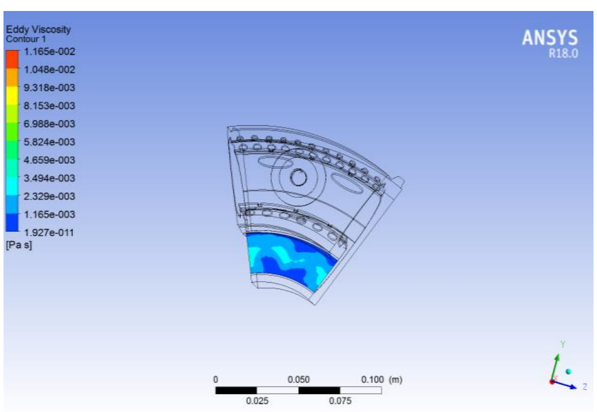

Figure 10: Eddy Viscosity profile at the outlet of the combustor

Table shows the output parameters obtained through numerical simulation of the combustor model and above distribution profiles

Table VI: Parameters obtained at the outlet

Sl. No.	Parameter	Unit	Values
1.	Air mass flow rate	Kg/s	0.816
4.	Total pressure	kPa	298
5	Total Temperature	K	1142
6.	Density	Kg/m^3	1.13
7.	Velocity	m/s	120.9
8.	Maximum Temperature	K	2498
9.	Pattern Factor	---	0.43
10.	Pressure Loss	%	2.8

VI. CONCLUSIONS

The design of the Reverse Flow Combustor and numerical investigations is carried out at basic design point operating conditions which shows a well-behaved flow with flame stabilization and equivalence ratio with in the design limits.

The Temperature distribution profile at the outlet is not uniform which suggest redesigning of the number and diameter of the air admission holes.

Exit gas analysis to be carried out for low NOx emissions at the design point conditions.

Further numerical simulation of the combustor at different speeds of the engine or mass flow rate of air to be carried out for performance evaluation of the designed combustor.

REFERENCES

[1] MELCONIAN, J.O; MODAK, A.T. Combustor design. In: SAWYER, JAW. (Ed.) Sawyer's gas turbine engineering handbook design. Volume 1, Theory & design. 3. ed. Connecticut: Turbo machinery International Publications, 1985. v.1, Chapter. 5, p 5-1-5.

[2] HODGEMAN, J.D.; MANIA, H; MOLARS, E.G. Assessment, development, and application of combustor aero thermal models. Washington, D.C: NASA, 1988. (NASA TM-100290)

[3] STUTTAFORD, P.J.; RUBINI P.A. Preliminary gas turbine combustor design using a network approach. ASME Journal of Engineering for Gas Turbines and Power, v.119, p.547-552. 1997.
[4] MELLOR, A. M.; FRITSKY, K. J. Turbine combustor preliminary design approach. Journal of Propulsion and Power, v.6, n.3, p.334-343, 1990.
[5] MONGIA, H.C. Perspective of combustion modeling for gas turbine combustors. In: AIAA AEROSPACE SCIENCES MEETING AND EXHIBIT, 42., 2004, Reno, Nevada. Proceedings... Washington, D.C: AIAA 2004. (AIAA-2004-156).
[6] LEFEBVRE, A.H. Gas turbine combustion. 2. ed. New York: Taylor & Francis, 1999.
[7] ODGERS, J. Combustion modelling within gas turbine engines, some applications and limitations. In: AEROSPACE SCIENCES MEETING, 1977, Los Angeles, California. Proceedings...Washington, D.C: AIAA 1977 (AIAA-1977-52).
[8] KNIGTH, H.A.; WALKER, R.B. The component pressure losses in combustion chambers. National gas turbine establishment Farnborough. Aeronautical research council, 1965. England. AD0029109.
[9] ALLAIRE, D.L. A physics-based emissions model for aircraft gas turbine combustors. 2006. 105p. Thesis (Master of Science-MSc) – Massachusetts Institute of Technology. Cambridge.
[10] TURNS, S. R. An introduction to combustion: concepts and applications. 2. ed. Boston: Mc Graw Hill. 2000.