CFD Modeling Of An Aero Gas Turbine Combustor For A Small Gas Turbine Engine

By

K. Sreenivasarao, Manager (Design), Aero Engine Research and Design Centre, Hindustan Aeronautics Limited, Bangalore-560093.
Prof. S.K. Bhatti, Department of Mechanical engineering, AU College of Engineering, Vishakapatnam.

Abstract

This paper discusses CFD applications to the analysis in small, high intensity complex flow field gas turbine combustors. Performance of combustor has been predicted using a commercially available CFD code ANSYS CFX. The entire flow field from the inlet to exit of the combustor, including the liner injection holes, has been modeled with reasonable accuracy. Comparison of the quality of temperature distribution and combustion efficiency at combustor exit are presented. A close agreement is observed between the predictions and experiments. Computational Fluid Dynamic analysis of 3D combustor flows and the empirical validation of results are used to demonstrate the effectiveness of CFD as a design tool in this challenging environment. It is demonstrated that commercial CFD codes provide qualitative and reasonable quantitative information that can be effectively used in the combustor design optimization

A reverse flow annular combustor for a small gas turbine engine have been modeled. The simulations were performed using commercial Computational fluid dynamics code ANSYS-CFX, in which a three-dimensional compressible k-ε turbulent flow model and a one-step overall chemical reaction between JET-A / air were used. Reverse flow annular combustor with simplex type of fuel injectors were used.

IJERT

Keywords: Aero Engine Annular, Design, Combustion Chamber, CFD, ANSYS CFX.

1.Introduction

The conventional approach to the design and development of combustion systems for gas turbine engines involves extensive use of empirical correlations, derived form experimental investigations and component development tests. The designer of a combustion system for a gas turbine engine is given the task of achieving the desired goals of the combustor, namely, complete combustion, low total pressure loss, proper temperature distribution at exit with no "hot spots", highly stable combustion, freedom from flameout, good re-light capability and operation over a wide range of mass flow rates, pressure and temperatures in small size.

The main objective of the present work is to select a suitable combustor for a small gas turbine engine using empirical correlations and a commercial CFD code CFX from ANSYS by modeling the complete geometry.

IJERT

2. Design of Annular Combustion Chamber

Saravanamutto et al. /19/ has given cycle analysis and performance characteristics of individual components of gas turbine engine. The data for designing Annular straight and reverse flow type combustion chambers is presented in Table I.

The design of the combustion chamber is carried out as outline in the literature /1,2,3,6,10,11,12,13,18/.

Table 2 gives the - dimensional details of combustion chamber designed. Figure 2 shows the sector models of combustion chambers.

Table-1: Design data for combustion chamber

S.No	Description	Value
1	Entry Pressure	10 bar
2	Entry temperature	640 K
3	Entry Air Flow	4.7 kg/s
4	Assumed pressure loss	5 %
5	Fuel / Air ratio	0.018786
6	Fuel flow	303 kg/hr
7	Turbine entry Temperature	1300 K
8	Limiting value of liner temperature	1000 K

Table-2: Geometrical parameters

S.No	Description	Value
1	Combustion chamber casing outer diameter (mm)	330 mm
2	Combustion chamber inner casing diameter (mm)	105 mm
3	Flame tube inner liner diameter (mm)	304 mm
4	Flame tube outer liner diameter (mm)	224 mm
5	Flame tube length (mm)	105 mm

Table-3: Air distribution holes

	Number of holes		dia (mm)
	inner	outer	
Dome cooling	200		0.4
Primary zone	48	80	5
Secondary zone	80	128	3.5
Dilution zone-1	2*144		2
Dilution zone-2	244	307	0.4
Liner cooling-1	1040	1008	0.6
Liner cooling-2	848		1

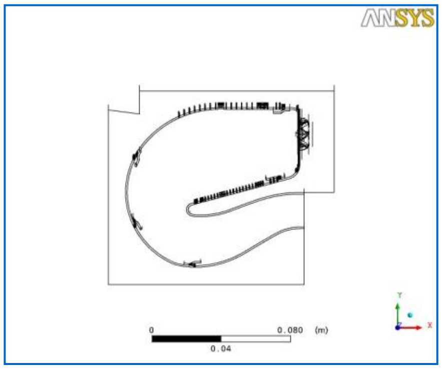

Fig.2: Reverse Flow Combustor Configuration

CFD Modeling Of The Combustor

The challenging task in aero gas turbine combustor CFD modeling is two-fold, namely, the development of models to describe the real world component geometries and an accurate description of the coupled interacting physical and chemical phenomena. An accurate modeling of the combustor geometry is required for arriving at realistic predictions. Initially, combustor simulations were limited to the flow field inside the liner and the diffuser / combustor annulus region. The solutions obtained by these simulations are strongly dependent on the accuracy levels associated with the description of boundary conditions. However, from the past few years, simulations are being carried out by these simulations are strongly dependent on the accuracy levels associated with the

IJERT

description of boundary conditions. However, from the past few years, simulations are being carried out by modeling the entire flow field from compressor exit to the turbine entry.

Physically phenomena that have to be addressed are turbulent transport, fuel spray atomization and vaporization and finite rate chemistry of the reactants. Based on the regime of operation of an aircraft engine, different components of these phenomena become more important while the others are less significant. Further, there are two main requisites for a correct evaluation of CFD models: the availability of a set of experimental data accurate and detailed enough to enable the specification of the boundary conditions, and a comparison between experimental and numerical values.

Grid Generation

A 22.5° Sector 3D model was generated by using UNIGRAPHICS. Model was meshed with around 30,00,000 tetrahedral elements by using ICEM TETRA.

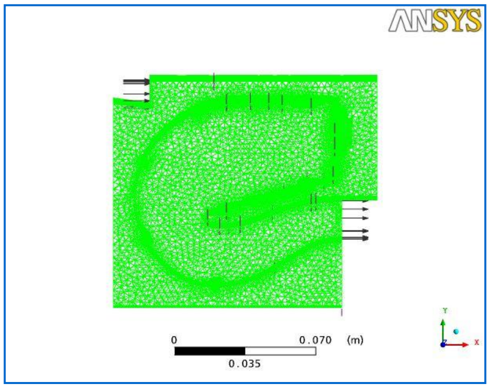

Fig.3: Mesh generation

The combustor considered in the present analysis is annular reverse flow with 16 swirlers equally spaced along the circumferential direction. Hence, the calculations are conducted for the flow in a 22.5° sector of the combustor with periodic boundary conditions being imposed on the two side planes. The outer and inner liners are provided with one row each of primary, secondary and dilution zones. In addition, the liners are cooled by effusion cooling. The air distribution through the liner holes is given in table-3

3.CFD Code

The commercially available CFD code ANSYS CFX is used for the analysis. ANSYS CFX solves a set of coupled transport equations for fluid, heat and mass transport processes in laminar or turbulent, compressible or incompressible, transient or steady and flows at any speed. It can also cater

IJERT

for chemical reactions, combustion, thermal radiation, soot formation, droplet transport, embedded moving boundaries, complex fluid property relations and a number of other fluid flow phenomena.

4.Boundary Conditions

The boundary conditions used for the analysis are given in Table-1, The simulation was performed using commercial Computational fluid dynamics code ANSYS-CFX, in which a three-dimensional compressible k-ε turbulent flow model and a one-step overall chemical reaction between JET-A / air was used. The fuel used in the present study is Jet-A aviation kerosene; It is assumed to be in a fully vaporized state and a single – phase calculation has been performed. The fuel is injected uniformly through an annular opening inside the swirler thus closely representing the actual atomizer. No slip condition is imposed at all the internal as well as the external walls of the computational domain.

Table-4: Boundary conditions

Inlet	Outlet	Fuel Inlet
Total Pressure = 10 bar	Mass flow rate of air = 4.73 kg/sec	Mass flow rate of fuel =0.73 kg/sec
Total Temperature = 640 K	-	Total Temperature =303 K

5.Numerical Simulations:

There are two ways to analyze the combustion chamber numerically. One way is to give input conditions at inlet and all the air admission holes as per the design conditions. But in actual case, the flow distribution in different zones" cannot be controlled. This is the biggest drawback of providing different inputs at different air admission holes.

The second way of analyzing the combustion chamber is to provide only one inlet at the inlet of the diffuser and let the flow divide by itself into liner and casing, and from casing into different zones through air admission holes and cooling slots. Such condition is the exact replica of the real case experimentation, in which the air is supplied at the inlet diffuser with known conditions of pressure, temperature and velocity, and then, allowed to divide between the casing and the

IJERT

liner with fuel injection at liner entry. Pressure swirl Atomizer is used for fuel injection in the present study. A three dimensional atomizer is modeled as shown in Figure 4.

6.Basic Assumptions

- Geometrical Approximations
- 3D model analysis
- Thickness of the sheet element is ignored

7.CFD Models

- 3D turbulent, steady and compressible
- Shear stress transport turbulence model
- Jet A liquid (C12 H23) as the fuel
- Eddy dissipation method for combustion modeling
- Solver – ANSYS CFX

CFD Modeling of Jet A liquid (C12 H23) Fuel

The combustor modeling is carried out using eddy dissipation model based on the concept that chemical reaction is fast relative to the transport processes in the flow. When reactants mix at the molecular level, they instantaneously form products.

The liquid evaporation model is a model for particles with heat transfer and one component of mass transfer, and in which the continuous gas phase is at a higher temperature than the particles. The model uses two mass transfer correlations depending on whether the droplet is above or below the boiling point. This is determined through the Antoine equation.

The LISA (Linearized Instability Sheet Atomization) model is able to simulate the effects of primary breakup in pressure-swirl atomizers. With pressure swirl injectors, the fuel is set into a rotational motion and the resulting centrifugal forces lead to a formation of a thin liquid film along the injector walls, surrounding an air core at the center of the injector. Outside the injection nozzle, the tangential motion of the fuel is transformed into a radial component and a liquid sheet is formed. This sheet is subject to aerodynamic instabilities that cause it to break up into ligaments. For secondary breakup, TAB (Taylor Analogy Breakup) Model is used. This spray breakup models will lead to drop diameter distribution depending upon the injection pressure differential (Atomizer model is shown in Figure 4).

IJERT

The exit conditions of Coefficient of discharge, drop diameter distribution, velocity and film thickness as obtained from CFD analysis of pressure swirl atomizer is given at a point in the present combustion chambers (Figure 5).

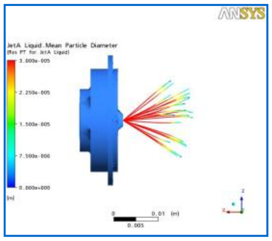

Fig.4: Atomizer model Fig.5: Atomized Liquid

8.Results and Discussions:

The quality of combustor exit temperature distribution is generally expressed in terms of non-dimensional parameter viz., Pattern factor, which can be defined as follows:

Pattern Factor = (T4max – T4avg) / (T4avg – T3avg)

The CFD analysis was carried out with defined boundary conditions described in previous section. The temperature and velocity distribution as found using Commercial CFD Code CFX for gas turbine combustion chambers designed at design condition are given in Figure 6 to Figure 9. Temperature profile and efficiency of combustion chamber are given in Figure 10 and 11. Number of configurations with varied hole distribution in the combustion liner were analyzed. Circumferential and Radial Pattern factors achieved 0.235 and 0.1. Combustor pressure loss achieved 6.52%.

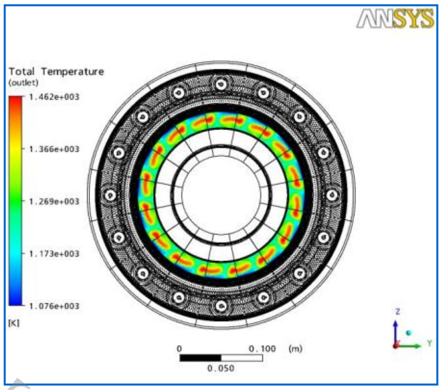

Fig.6: Total Temperature Distribution at outlet

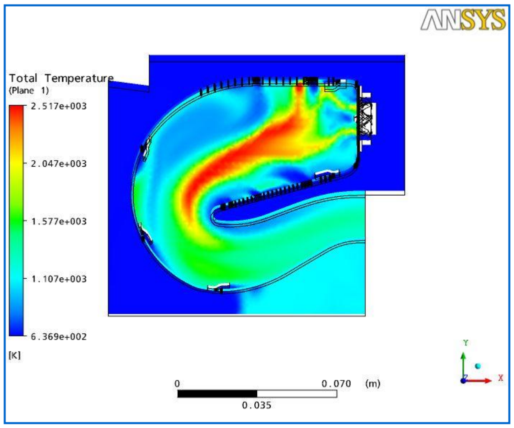

Fig.7: Total Temperature Distribution On XY Plane

IJERT

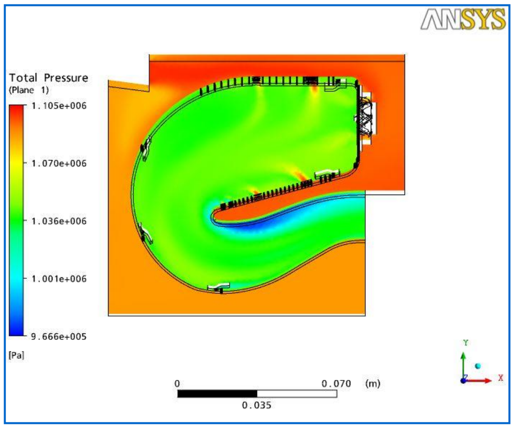

Fig.8: Total Pressure Distribution On XY Plane

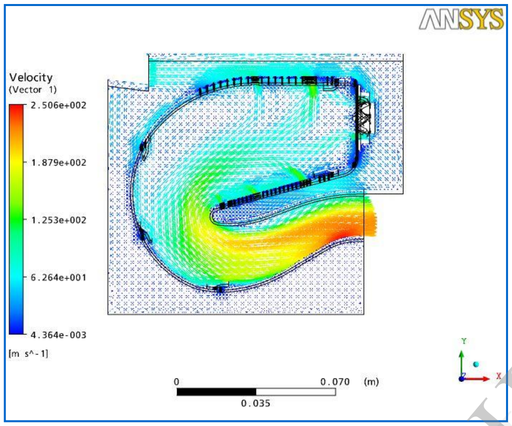

Fig.9: Velocity Vector Distribution On XY Plane

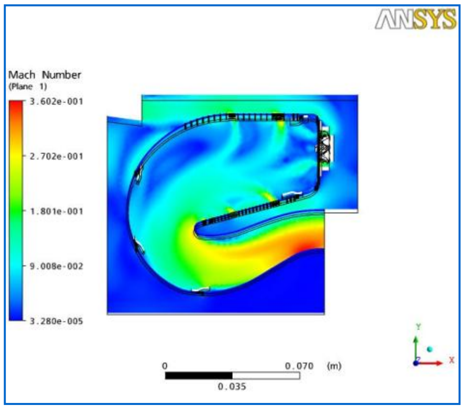

Fig.9: Mach number variation On XY Plane

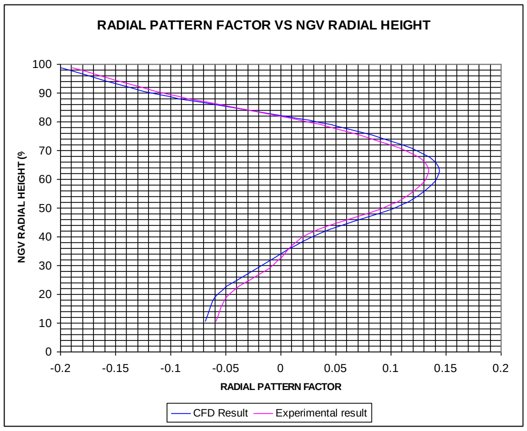

Fig.10: Temperature profile distribution

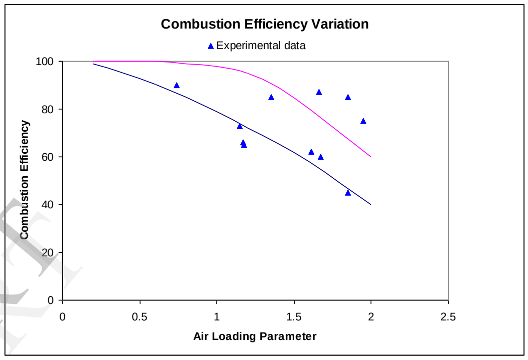

Fig.10: Combustion efficiency variation

IJERT

9. References

1. Lefebvre A. H., Gas Turbine Combustion, Taylor and Francis, 2nd ed., 1999 ISBN 1 - 56032 -673 - 5.

2. Frankenberger E. & Black-Schaub P., Modeling Design Process in Industry – Empirical Investigations of Design Work in Practice, Automation in Construction, Vol. 7, pp. 139 - 155,1998.

3. Rogers J M, A Generic Algorithms based Optimization Tool for the Preliminary Design of Gas Turbine Combustors, Ph. D. Thesis, School of

Mechanical Engineering, Cranfield University, UK, 2003.

4. Anand M. S. & Priddin C. H., Combustion CFD - A Key Driver to Reduce Development Cost and Time, Fifteenth International Symposium on Air Breathing Engines", Bangalore, India, ISABE - 2001 - 1087, 2001.

5. D. Caraeni, C. Bergstrom & L. Fuchs, Modeling of Liquid Fuel Injection, Evaporation and Mixing in a Gas Turbine Burner Using Large Eddy Simulations, Flow, Turbulence and Combustion, Vol. 65, pp. 223 - 244,2000

6. Philippe Dagaut & Michel Cathonnet, The Ignition, Oxidation and Combustion of Kerosene: A Review of Experimental and Kinetic Modeling, Progress in Energy and Combustion Science, Vol. 32, pp. 48-92, 2006.

7. F.Z. Sierra, 1. Kubiak, G. Gonzalez & G. Urquiza, Predictions of Temperature Front in a Gas Turbine Combustion Chamber, Journ. Applied Thermal Engineering, Vol. 25, pp: 1127-1140,2005.

8. B. Zamuner, P. Gilbank, D. Bissieres and C. Berat, Numerical Simulation of the Reactive Two-Phase Flow in A Kerosene/Air Tubular Combustor, Aerospace Science and Technology, Vol. 6, pp. 521-529,2002.

9. F. F. Grinstein & c. Fureby, LES Studies of The Flow in A Swirl Gas Combustor Proceedings of the Combustion Institute, Vol. 30, pp. 179 1-1798,2005

10. Mellor A. M., Design of Modern Turbine Combustors, ACADEMIC PRESS INC, 1990.

11. A. H. Lefebvre and E. R. Norster, A Design of Tubular Combustion Chambers for optimum Mixing Performance, Technical Advances in Gas Turbine Design, pt. 3N, Proc. Inst. Mech. Eng., 1969.

12. K. S. Kaddah, Discharge Coefficients and Jet Discharge Angles for Combustor Liner Air Entry Holes, Website of Cranfield, England.

13. Norster E. R, Jet Penetration and Mixing, Cranfield University, England, 1964.

14. I. R. Gran, M. C. Melaaen and B. F. Magnussen, Numerical Simulation of Local Extinction Effects in Turbulent Combustor Flows of Methane and Air, Twenty-Fifth Symposium (International) on Combustion, Combustion Institute, pp. 1283-1291, 1994

15. W. Shyy, S. M. Correa and M. E. Braaten, Computation of Flow in a Gas Turbine Combustor, Combustion Sci. & Tech., Vol. 58, pp. 97 -1 17,1988

16. W. Shyy, M. E. Braaten & D. L. Burrus, Study of Three-Dimensional Gas-Turbine Combustor Flows, Int. 1. Heat and Mass Transfer. Vol. 32, No.6, pp. 1155-1164, 1989.

IJERT

17. Charles K. Westbrook, Yasuhiro Mizobuchi, Thierry 1. Poinsot, Phillip 1. Smith & Jurgen Warnatz, Computational Combustion, Proceeding of Combustion Institute,Vol. 30, pp. 125 - 157,2005
18. Kulshreshtha D. B. & Channiwala S. A., Simplified Design of Combustion Chamber for Small Gas Turbine Applications, Proc. Of International Mechanical Engineering Congress and Exposition 2005, Paper Code IMECE2005 - 79038.
19. Saravanamutto H, Cohen H, and Rogers G, Gas turbine theory, 6th ed., ISBN 978 - 0 - 13 - 22437 - 6, 2008

IJERT