Large Turbulence Creation Inside A Gas Turbine Combustion Chamber Using Perforated Plate

S. R. Dhineshkumar[1], B. Prakash2, S. R. Balakrishnan[3]
PG scholar[1], Wing commander (Retd)[2], Director/H.O.D[3]
[1, 2,3]*Department of Aeronautical Engineering, Nehru institute of engineering and technology*

Abstract

This project documents the development of a new design algorithm for a modern gas turbine combustor. The algorithm includes a set of preliminary design procedures involving the use of perforated plate models. The perforated plate model captures complex processes such as, chemical reaction, jet mixing, and produces more turbulence .Due to the turbulence the fuel mixes with air thoroughly and gives better combustion efficiency. The preliminary design procedures are verified using the advanced numerical techniques of computational fluid dynamics (CFD). These techniques are used to solve the swirling flow field inside the pre-mixer, the reacting flow field inside the liner, and the rate of turbulence level inside a combustion chamber. A three-dimensional solid model of the combustor and a complete set of engineering drawings were prepared using CATIA V5.

Keywords— *Combustor, Perforated plate, Turbulence, Swirling flow, Combustion efficiency*

1. Introduction

The modern combustor designs have emerged to achieve emission requirements while maintaining the high combustion efficiency and good flame stability characteristics of conventional combustors. Since published design methodologies for conventional combustors do not apply well to these modern designs, and current designs of these are typically regarded as proprietary, there is a need for the development of new design methodologies, particularly for lean premixed combustors. This project documents the development of design methods and then applies them to a 1-MW gas turbine engine.

Designs use one or a combination of several concepts: axial or radial staging, variable geometry, and premixing. The stability of a flame is characterized by the ability to burn steadily at a fixed position without blowing out. Aircraft engines have strict stability requirements and therefore have only incorporated variable geometry with conventional combustor design to avoid any unacceptable flame blowout over a wide range of combustion air/fuel ratios. In this paper, the design uses premixing concept in addition with creating recirculation for rapid mixing of air and fuel

2. Premixer

Pre-mixers play an important role in modern combustors. Pre-mixers are devices composed of one or more swirlers designed to mix the fuel and air prior to combustion, as shown in Figure 1.This project shows that the Swirler used in the modern premixer design is replaced with the perforated plate design. Design must also ensure that the fuel/air mixture does not reside in the pre-mixer for too long and auto ignites. The mixture must also move fast enough to ensure that flashback does not occur.

The design and performance of a combustor is strongly affected by aerodynamic processes (Lefebvre, 1999). The performance of perforated plate designs differs mainly due to the aerodynamic efficiency, a measure of the effectiveness at introducing and distributing air in the premixer (Scull & Mickelsen, 1957). The achievement of aerodynamically efficient designs, characterized by good mixing and stable flow patterns with minimal parasitic losses, is one of the primary design objectives.

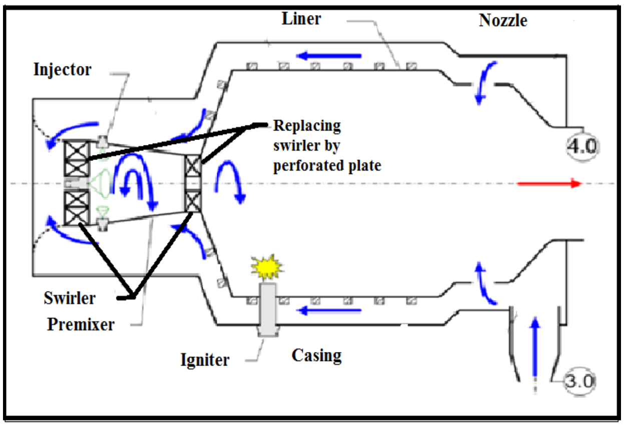

Fig. 1 Combustor with Perforated plate replacing Swirler

3. Central Re-circulation

The perforated plate produces a central recirculation by which Fuel and air must move slowly enough for the flame to propagate upstream and ignite fresh mixture. The point at which the flame can no longer propagate back through the flow is the stabilization point or anchor. Zones of flow reversal help stabilize the flame by creating localized regions of low velocity flow called flame holders. The design introduced in this paper also act as a partial flame holder. Large scale central recirculation zones serve many other purposes as well. Hot combustion products become trapped in the recirculation mass and are returned to the combustor dome inlet. This hot gas helps stabilize the flame by providing a continual source of ignition to the incoming fuel. It also serves as a zone of intense mixing within the combustor by promoting turbulence through high levels of shear between the forward and reverse flows.

Central recirculation forms at the onset of flow reversal when an adverse axial pressure gradient exceeds the kinetic energy of the incoming flow (Beer & Chigier, 1972). Adverse pressure gradients may be introduced by creating high degrees of swirl or angular momentum at the inlet of the combustor and large sudden expansions in areas such as dumps or bluff bodies

4. Perforated plate

Perforated plates are static mixing devices used to impart swirl to the flow. The goal of perforated plate design is to maximize the benefits of recirculation by imparting sufficient turbulence to the flow while minimizing the incurred pressure losses. Modern combustors also use swirlers to promote mixing of the fuel and air in the pre-mixer prior to combustion. There are many methods of producing swirl. These include axial, radial, tangential, and discrete jet swirlers. In this paper a new perforated plate design method is included for producing swirl.

4.1 Preliminary design

The preliminary design procedures were implemented in a analysis program and applied to a 1 MW gas turbine engine. The application necessitated many decisions made based on the engine specifications and requirements. In order to formulate a preliminary design, a compromise must be made between the convenience of simple algorithms and the accuracy of complex numerical models that require vastly larger amounts of computing resources.

5. Concept

Design begins with a concept. Figure 1 illustrates the concept chosen. Air discharged from the compressor enters the combustor at Station 3.0. The air passes through a faired pre-diffuser before it is dumped into the annulus. In this region, just downstream of the sudden expansion, the cross-sectional flow area in the annulus is large to ensure that any asymmetry in the flow is avoided. As the air accelerates through the annulus, a portion is admitted by holes in the liner to dilute the hot combustion gases. The other portion of the air flows through the annulus where it cools the outside of the liner wall. This cooling effect is enhanced by the use of trip strips. Annulus air used for cooling is then dumped into a plenum and enters the pre-mixer. Inside the pre-mixer, the air passes through the perforated plates having holes of 20 mm diameter facing opposite to the injector to mix with an evaporating liquid fuel spray. The exiting fuel and air mixture is dumped into the combustor PZ by another perforated plate where it ignites and burns. The resulting hot products are diluted with relatively cooler air and accelerated out of the combustor by a converging nozzle.

5.1 Perforated plate Dimensions

The hub at the centre of the perforated plate was sized so that a fuel nozzle maybe mounted inside.

Dhub,1 = 155 mm A slightly larger value was used for the combustor Swirler hub.

Dhub,2 = 144 mm

Perforated plate holes are made with

D_{out} hole = 20mm at 30^0

D_{in} hole = 20mm at 30^0

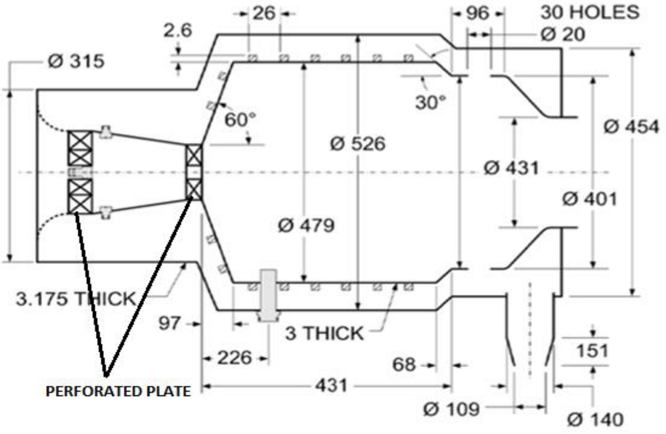

Fig 1 Overall Combustion Chamber Dimensions

6. Verification of preliminary design

To assess the success of the preliminary design procedure, the aerodynamic performance of key components was verified using computational fluid dynamics (CFD). CFD refers to the application of any numerical technique to fluid flow. The preliminary design procedure formed was verified with CFD simulation of the perforated plate, CFD simulation of the combustor, and Analyses of flow properties with CFD results

6.1 Geometry And Grid Generation

A simplified three-dimensional solid model has been built and used to generate the Computational grid

6.2 Solid model

The internal flow path of the premixer with perforated plate was modelled using some basic simplifications. This includes that No swirlers were included in the model. The flow inside the premixer is uncoupled from the flow in the annulus. Only the inside of the premixer was modelled due to the complexity of the combustor flow path and limitations of computational resources. The problem is axisymmetric. A perforated plate section was modelled using the periodic boundary condition to reduce the grid size and computation time.

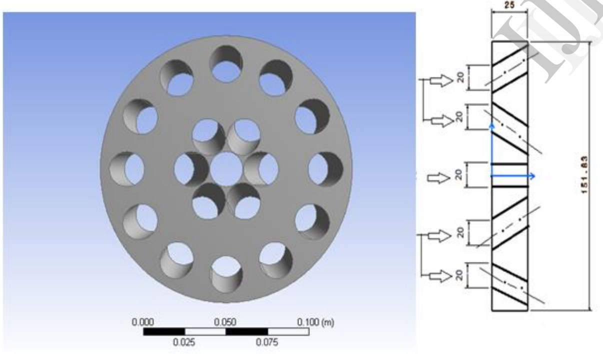

Fig 2 Solid model of perforated plate flow domain and Dimensions of Perforated Plate

6.3 Grid Generation

An unstructured grid was generated with ANSYS CFX-MESH, which consists of 15771 nodes and 76967elements. The nodal density of the mesh was selected by studying its effects on the overall solution and choosing one whose solution was grid independent

Turbulence Model	k-ϵ
Continuous Phase Heat Transfer Model	Total Energy
Discrete Phase Heat Transfer Model	Particle Temperature
Particle Coupling	Fully Coupled
Drag Force	Schiller Naumann
Turbulent Dispersion Force	None
Heat Transfer	Ranz Marshall

Table 1. Fluid model for premixer and combustor

7. Boundary conditions for premixer and combustor

The appropriate choice of boundary conditions is essential and is a critical part in modelling a flow accurately. Typical boundary conditions for FLUENT simulation were the inlet, the wall and the outlet boundaries

7.1 Inlet

At the inlet of the computational region, the inlet boundary condition is defined as mass flow inlet. Some assumptions about boundary conditions that were not directly measured had to be made as follows:

Condition	**Value**
Flow regime	Subsonic
Mass flow rate	0.471 kg/s
Flow direction	Normal to boundary
Turbulent viscosity ratio	10%
Temperature	473 K
Velocity	40 m/s

Table 2.Inlet Boundary Condition

7.2 Outlet

The exit boundary is defined as pressure outlet .The average static pressure was set to match the inlet total pressure with that predicted by the preliminary design. The details of the outlet boundary condition are provided in Table 3

Condition	**Value**
Flow regime	Subsonic
Outlet pressure	4.4 bar
Backward Turbulence intensity	0.01%
Backward Turbulent viscosity ratio	10%

Table 3.Outlet boundary condition

7.3 Wall Boundary

Wall boundary conditions were placed on both the perforated plate and the premixer wall. The perforated plate was modelled as an adiabatic wall. Their details are listed in Tables 4

Condition	**Value**
Wall influence on flow	No slip
Wall roughness	Smooth wall
Heat transfer	Adiabatic
Emissivity	0.8
Diffuse fraction	1

Table 4. Wall Boundary

7.4 Velocity Flow Field

The velocity flow field is depicted in Figure 5.5. A small undesirable central recirculation bubble is observed behind the plate as a result of the strong swirling flow. However, is too low for combustion too occurs here. Nonetheless, the recirculation zone suggests that the axial velocity of the flow issuing from the inner mixer plate is not high enough to hinder reverse flow along the centre body. The velocity flow angle of the air inside the premixer at various axial locations, labelled in Figure 5.5, was studied to determine the strength of swirl exiting the premixer. Figure 5.7 illustrates that the tangential velocity of the flow degrades due to the shear layer formed between the two opposite facing streams. The figure supports the assumption that a second plate is necessary at the exit of the premixer to impart sufficient swirl for PZ recirculation.

7.5 Turbulence Flow Field

The turbulent kinetic energy field has shown exhibits different high intensity regions inside the injector. Swirl region, formed by the eighteen holes of swirling perforated plate, includes the head-end region near the centreline, where the liquid fuel discharged.

7.6 Path Lines

The figure predicts the motion of air particles inside the premixer. This shows the recirculation clearly created by the holes of the perforated plate

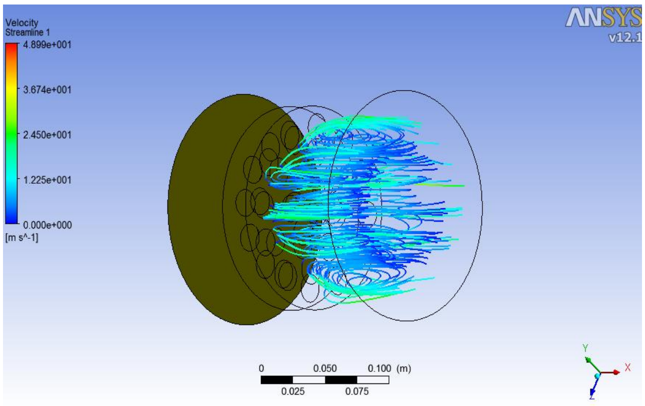

Fig 5. Path Line Followed By Fluid

8. Combustor Domain

CFD analysis was performed to measure the performance of the combustor at the design point with respect to mixing and evaporation. This section outlines the steps taken to perform the CFD simulation and discusses the results. The steps include generating a modelling domain and grid, specifying the boundary conditions, setting up the solver, and solving the domain.

8.1 Solid model

The combustor flow domain was simplified to reduce the complexity of the problem. The simplifications includes that, no swirlers were included in the model. The size of the mesh was vastly reduced by placing the inlet to the domain upstream of the mixer plate. The main inlet is placed upstream of the premixer perforated plate. It is reasonable to assume that the flow exiting this perforated is turbulent and that it follows into the combustor.

8.2 Grid Generation

The computational grid generated from the solid model is illustrated in Figure. The unstructured grid with 125,000 nodes and 690,000 elements was refined at the hub and thin wall to help capture the large velocity gradients expected

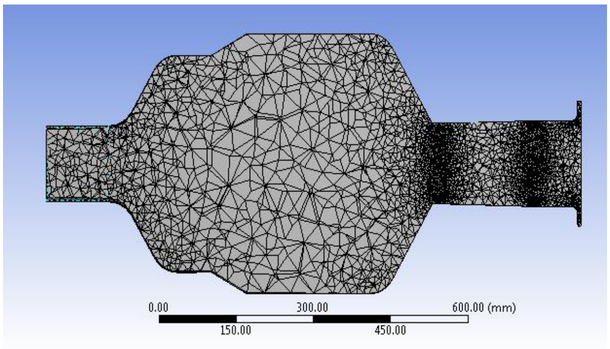

Fig. 7 Computational Solid Model Grid

8.3 Velocity Flow Field

The velocity inside the combustor was contoured to verify that a strong swirling flow exists depicts a large torroidal recirculation zone at the centre of the combustor .The type of recirculation zone observed is typical of strongly swirling flows where the induced axial pressure gradient is strong enough to force the flow to stick to the outer walls (Figure 5.9). A second recirculation zone is observed in the aft portion of the second perforated plate, upstream of the dilution holes. The recirculation incurs additional pressure losses that are considered parasitic as they do not contribute to the overall combustion process. It is an unavoidable result of the sudden contraction at the exit of the liner.

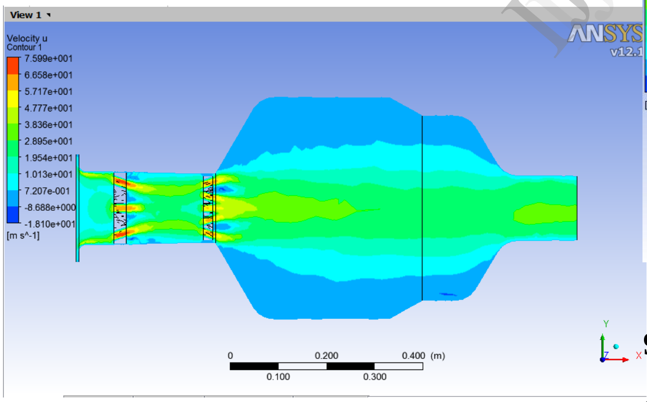

Fig 6. Velocity Flow field

8.4 Turbulence Flow Field

The fig shows the field surrounding the central recirculation zone in the main flow passage. Strong shear layers develop, especially when the primary swirling flow merges with the counter-opposing flow through the injectors of secondary perforated plate. The strong shear force associated with the counter-opposing flows in the opposite direction also enhances the atomization process. Thus, the flow structure in the present combustor provides an effective capability to atomize the liquid fuel.

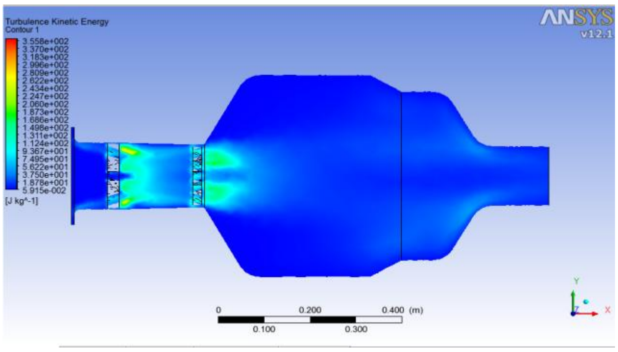

Fig.8 Turbulence Kinetic Flow Field

8.5 Pressure Flow Field

The re-circulation zone is formed due to the effect of the radial pressure gradient created by the perforated plate and also due to the interaction of opposing primary jets, which have been modelled carefully. This re-circulation zone is very important for flame stabilization in the primary zone so that the flame remains anchored to the atomizer and almost complete burning must take place in the primary zone to get a good exit pattern factor.

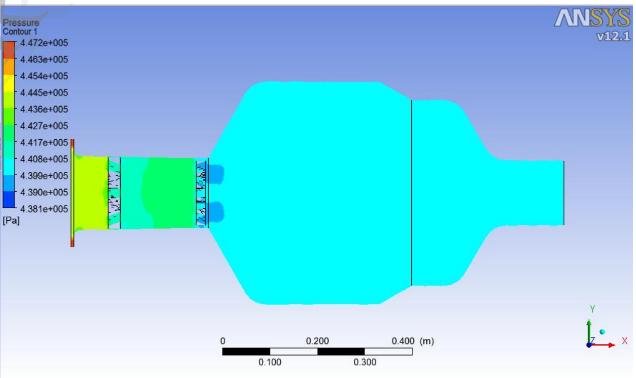

Fig. 9 Pressure Flow Field

9. Conclusions

Based on a preliminary design of the perforated plate the analysis of flow properties inside a combustion chamber after replacing the swirlers by a perforated plate was carried out. The perforated plate employed in this research show a significant effect on the flow pattern within the combustor model the first perforated plate produced a small volume of recirculation zone while 2nd perforated plate produces larger recirculation zone size. From the parametric study carried out with various angle of holes in perforated plate, it is found that 30° holed perforated plate is the best for producing appropriate recirculation zone with reasonable pressure drop. This perforated plate is reasonably performing with the results of Swirler.

10. References

[1] Li, G., & Gutmark, E.J. 2004. *Effects of Swirler Configurations on Flow Structures and Combustion Characteristics.* ASME Paper GT2004-53674

[2] Lefebvre, A.H. 1989. Atomization and Sprays. New York: Hemisphere Publishing

[3] Lefebvre, A.H. 1984. *Fuel Effects on Gas Turbine Combustion* - Liner Temperature, Pattern Factor, and Pollutant Emissions. AIAA Journal of Aircraft, 21(11), 887–[5]898

[4] Scull, W.E., & Mickelsen, W.R. 1957. *Flow and Mixing Processes in Combustion Chambers*. Chap. II, pages 226–276 of: Basic Considerations in the Combustion of Hydrocarbon Fuels with Air. National Advisory Committee for Aeronautics. Report 1300.

[5] Beer, J.M., & Chigier, N.A. 1972. *Combustion Aerodynamics.* London: Applied Science Publishers LTD.

[6] Saravanamuttoo, H.I.H., Rogers, G.F.C., & Cohen, H. 2001. '*Gas Turbine Theory'*. 5edn. New York: Prentice Hall.

[7] Dodds, W.J., & Bahr, D.W. 1990. *Combustion System Design.* Chap. 4, pages 343–476 of: Mellor, A.M. (ed). '*Design of Modern Turbine Combustors*'. New York:Academic Press.

[8] Chin, J.S., & Lefebvre, A.H. 1982. '*Effective Values of Evaporation Constant for Hydrocarbon Fuel Drops*'. Pages 325–331 of: Proceedings of the 20th Automotive Technology Development Contractor Coordination Meeting.

[9] Gauthier, J.E.D. 2003. '*Gas Turbines'*. Carleton University, Ottawa. Lecture Notes for MECH 5402.

[10] Conference on Turbulence and Interactions TI2006, Porquerolles, FranceD Veynante'*large eddy simulations of turbulent Combustion*' (May 29 - June 2, 2006)

[11] Sang Cheol Ko and Hyung Jin Sung† (Received 18 July 2001)*Large-eddy simulation of turbulent flow inside a sudden-expansion cylindrical chamber*'

[12] Sourabh V. Apte, Oregon State University, 204 Rogers Hall, Corvallis, OR 97331,USA, (11 June 2008).'*Large-Eddy Simulation of Evaporating Spray in a Coaxial Combustor'*

[13] Simon Mendez1, Franck Nicoud1, and Thierry Poinsot1 *'Large-Eddy Simulation of a Turbulent Flow around a Multi-Perforated Plate'*

[14] S. Apte,G. Iaccarino,F. Ham,P. Moin Centre for Integrated Turbulence Simulations, Stanford University, Building 500,Stanford, CA 94305 *Large-Eddy Simulation of Reacting Turbulent Flows in Complex Geometries*'

[15] Bissières, C. Bérat DT/CP/CTE, TURBOMECA, 64511 Bordes Cedex, France,'*Large Eddy Simulation predictions and validations of a gas turbine combustion chamber*

[16] Centre for Integrated Turbulence Simulations Stanford University, Stanford, CA '*Large-eddy Simulation of Realistic Gas Turbine Combustors'* (December 1, 2005)