Propulsion and Power Research 2016;5(2):97–107

HOSTED BY

http://ppr.buaa.edu.cn/

Propulsion and Power Research

www.sciencedirect.com

ORIGINAL ARTICLE

Design and analysis of annular combustion chamber of a low bypass turbofan engine in a jet trainer aircraft

C. Priyant Mark[a],*, A. Selwyn[b]

[a]*Department of Aerospace Engineering, Karunya University, Coimbatore, T.N., India*
[b]*A.E.R.D.C., Hindustan Aeronautics Ltd., Bangalore, KA, India*

Received 19 May 2014; accepted 4 March 2016
Available online 26 May 2016

KEYWORDS

Aerodynamic design;
Annular combustion chamber;
CFD (computational fluid dynamics) analysis;
Gas turbine engine;
Optimization;
Real-time model

Abstract The design of an annular combustion chamber in a gas turbine engine is the backbone of this paper. It is specifically designed for a low bypass turbofan engine in a jet trainer aircraft. The combustion chamber is positioned in between the compressor and turbine. It has to be designed based on the constant pressure, enthalpy addition process. The present methodology deals with the computation of the initial design parameters from benchmarking of real-time industry standards and arriving at optimized values. It is then studied for feasibility and finalized. Then the various dimensions of the combustor are calculated based on different empirical formulas. The air mass flow is then distributed across the zones of the combustor. The cooling requirement is met using the cooling holes. Finally the variations of parameters at different points are calculated. The whole combustion chamber is modeled using Siemens NX 8.0, a modeling software and presented. The model is then analyzed using various parameters at various stages and levels to determine the optimized design. The aerodynamic flow characteristics is simulated numerically by means of ANSYS 14.5 software suite. The air-fuel mixture, combustion-turbulence, thermal and cooling analysis is carried out. The analysis is performed at various scenarios and compared. The results are then presented in image outputs and graphs.

*Corresponding author.
E-mail address: priyant.mark.c@gmail.com (C. Priyant Mark).
Peer review under responsibility of National Laboratory for Aeronautics and Astronautics, China.

http://dx.doi.org/10.1016/j.jppr.2016.04.001

1. Introduction

Gas turbine engine evolved as a critical part and the most efficient propulsion unit for aircrafts. It is now used in almost all of the passenger aircrafts worldwide with different variations. Military aircraft made the debut in using the turbojet engine. As the technology progressed, high performance military aircraft began using low bypass turbofan engines due to its advanced capabilities, efficiency and reliability, even at supersonic speeds.

Low bypass turbofans have a bypass ratio of around 1:1 or less [1]. A high specific thrust/low bypass ratio turbofan normally has a multi-stage fan, developing a relatively high pressure ratio and, thus, yielding a high (mixed or cold) exhaust velocity. The core airflow needs to be large enough to give sufficient core power to drive the fan. A smaller core flow/higher bypass ratio cycle (for the fan operation) can be achieved by raising the high pressure (HP) turbine rotor inlet temperature. The temperature rise of the airflow from the intake to the nozzle of the engine is also less, which results in a reduced fuel flow leading to a better specific fuel consumption (SFC) for the same pressure ratio. Thus, a low bypass turbo fan would add to the efficiency of the engine.

Jet fighters as well as trainers are high performance aircraft that use the most powerful engines for producing thrust. The process of upgrading military hardware has initiated the race to develop even more powerful engines. By increasing power, the engines require more fuel input, thereby resulting in fuel guzzling engines. This directly points to an inefficient engine in terms of fuel consumption. Fuel consumption efficiency is required even in military aircraft as it can aid in increasing the range. For improving efficiency, the very fundamentals lie in the combustion chamber. An efficient combustion chamber is the answer for better performance.

The most commonly used type of combustor is the fully annular combustor, the others being tubular and tuboannular combustor. Annular combustors [2] do away with the separate combustion zones and simply have a continuous liner and casing in a ring (the annulus). There are many advantages to annular combustors, including more uniform combustion, shorter size (therefore lighter), and less surface area. Additionally, annular combustors tend to have very uniform exit temperatures. They also have the lowest pressure drop of the three designs (on the order of 5%). The annular design is also simpler, although testing generally requires a full size test rig. Most modern engines use annular combustors; likewise, most combustor research and development focuses on improving this type. This paper deals with designing an efficient annular combustion chamber for use in jet trainer aircrafts.

Conrado [3] has discussed a design methodology which follows a similar approach for designing a micro gas turbine combustor. It also showcases an example and further automating the same approach using a computer program for ease of use. Silva [4] has discussed a consolidated design methodology for an automotive turbocharger utilizing a micro gas turbine combustor. It gives a brief report along with heat-transfer analysis. Generally, the computational fluid dynamics (CFD) analysis of a combustor is carried out based on different combustion models [5]. Few models such as Westbroor-Dryer one step model and Westbroor-Dryer two step model hold good for laminar combustion simulation. Likewise K-epsilon model, K-omega model and K-omega shear stress transport (SST) model hold good for turbulent combustion simulation. The present paper discusses mainly about designing a gas turbine combustor at a scale of a jet trainer aircraft engine using the most straightforward and transparent approach. It also focuses on reducing the development time and gives ample support for refining the design at every phase. The paper also presents a computer aided design (CAD) model designed using the same principles to show the practicality in using the design. For an accurate CFD analysis result of a gas turbine combustion chamber, it needs to simulate combustion and turbulence simultaneously. This paper gives a detailed CFD analysis report of the designed combustor based on the combustion-turbulence interaction model.

2. Aerodynamic design

2.1. Preliminary design procedure

The procedure purposed by Melconian and Modak (1985) [6] to design a combustor is described in Figure 1. The equations utilized in the design procedure is presented, which is sufficient for the reader to understand the design methodology idea.

2.2. Initial design parameters

The initial design parameters are mostly the compressor exit and turbine inlet constraints, which is usually absorbed for any combustion chamber design. Others include customer specifications, constants, experimental values and limits. Table 1 shows the initial parameters used for the design, which were obtained from real-time data.

3. Dimensions

3.1. Casing area

Eq. (1) calculates the reference area [7].

$$A_{ref} = \left[\frac{R}{2} \left(\frac{\dot{m}_3 T_3^{0.5}}{P_3} \right)^2 \frac{\Delta P_{3-4}}{q_{ref}} \left(\frac{\Delta P_{3-4}}{P_3} \right)^{-1} \right]^{0.5} \tag{1}$$