Applied Thermal Engineering 90 (2015) 722–732

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Research paper

The unsteady turbulence flow of cold and combustion case in different trapped vortex combustor

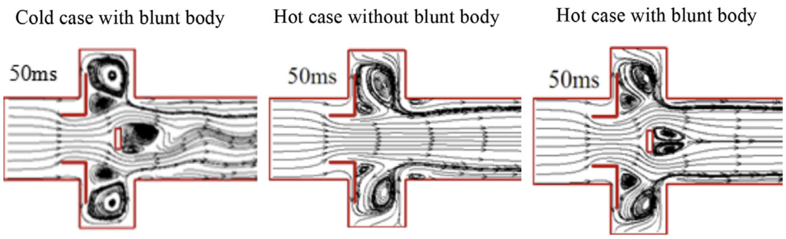
Zhuoxiong Zeng [a, *], Jianxing Ren [a], Xiaojing Liu [a], Zhou Xu [b]

[a] College of Power and Mechanical Engineering, Shanghai University of Electric Power, Shanghai 200090, China
[b] School of Aircraft Engineering, Nanchang Hangkong University, Nanchang 330063, China

HIGHLIGHTS

- A new structure of TVC with flow guide vane and blunt body was proposed.
- The effect of blunt body on the flow combustion characters was investigated.
- The flow field of combustor with blunt body and without blunt body is compared.
- Vortex undergoes the process of forming-breaking down-forming again-being stable.

GRAPHICAL ABSTRACT



ARTICLE INFO

Article history:
Received 29 April 2015
Accepted 20 July 2015
Available online 28 July 2015

Keywords:
Guide vane
Blunt body
Trapped vortex combustor
Unsteady flow

ABSTRACT

The trapped vortex combustor (TVC) is a new design concept in which cavities are designed to trap a vortex flow structure. In order to improve combustion efficiency of trapped vortex combustor, a new structure with guide vane and blunt body is proposed. Numerical simulation of unsteady turbulence flow is carried out, and the flow field of different combustor with blunt body and without blunt body is compared to analyze the effect brought by blunt body. The results show that the setting of blunt body can increase the value and the distribution area of turbulent kinetic energy, and improve the heat and mass transfer of the combustion chamber. Double vortex structure, no matter size or location, basically remains unchanged since it is formed, setting blunt body will not affect the cavity flow. Furthermore, after setting blunt body, the combustion chamber exit temperature distribution is more uniform, combustion efficiency is higher.

1. Introduction

A new combustion chamber-TVC (trapped vortex combustor) was proposed in the 1990s [1], its main contribution is to verify the idea of flame stability by cavity vortex. Because of the simple structure, small pressure loss, good flame stability and high combustion efficiency, etc, the model attracts more and more attention.

Chaouki [2] performed numerical analysis to test the combustion performance and emissions from TVC when natural gas fuel (methane) is replaced with hydrogen and synthetic gas (syngas). The results show that the flame for methane combustion is located in the primary vortex region but it is shifted to the secondary vortex region for hydrogen combustion. Ezhil [3,4] revealed that a single vortex is formed in the cavity for the low momentum flux ratio(MFR) case (MFR ~0.57). But for the higher MFR cases (MFR

---

* Corresponding author.
E-mail address: zengzhx@163.com (Z. Zeng).

http://dx.doi.org/10.1016/j.applthermaleng.2015.07.041
1359-4311/

~0.82 and 1.8), the flow and flame structures are altered considerably. The momentum flux ratio can play an important role in altering the flow and flame structures within the TVC cavity. Singhal [5] studied the single cavity TVC dynamics, the results indicates that reducing length-to-depth (L/D) ratio and increasing cavity-air velocity favor stable combustion. For certain cases, the flame tends to blowout whereas at higher and lower cavity-air velocities, the flame is observed to be stable. Vengadesan [6] performed numerical simulation involving passive flow through the two-cavity TVC to obtain an optimum cavity size to trap stable vortices inside the second cavity and to observe the characteristics of the two cavity TVC. Selvaganesh [7] revealed that the primary air needs to be injected for accommodating the decrease in oxidizer inside the cavity to obtain better performance from the TVC. Mishra [8] found the fact that, for a given length-to-depth (L/D), there is an optimal momentum ratio between the cavity injections and mainstream flow, which results in a single large vortex being trapped that can roll up the fuel-air injected and distribute them across the cavity. Blunck [9] used a trapped-vortex ultra-compact combustor (UCC) to study the effect of inlet air distribution, air loading, and turning vanes on combustion efficiencies, exit temperature profiles, and NOx emissions. Well-developed vortical structures enhance mixing between the fuel and air and increase the residence time. Jin [10] placed emphasis on vortical flow pattern within cavities, turbulence intensity distribution and interaction between cavity stream and mainstream. Flame images at different operating conditions are obtained and the effects of fuel to air ratio (FAR) and inlet Mach number on the physical appearance of flames are clearly revealed. Jin [11] investigated the effect of the cavity-injector/radial-strut relative position on the performance of a TVC. Xing [12] investigated the lean blow-out (LBO) performance of several combustors which utilized trapped vortex in the cavity to improve the flame stability. Zhang [13] studied the influence of the mainstream air flow velocity, the air intake velocity in the cavity, the height of inlet channel, the structure of holder and the structural proportion of the cavity on entrainment in the cavity.

In above, there are two type TVC structure primarily: single vortex structure and double vortex structure. For a single vortex structure, this TVC can form trapped vortex in cavity, provide heat for the mainstream and play the role of flame stability, but it is difficult to achieve rapid and uniform mixing of high temperature fired gas and the mainstream, which affects the fast ignition and limits to improve the combustion efficiency. For double vortex structure, the gas is sprayed into the cavity from forewall bottom and afterwall middle position, one large and one small vortex are formed in the cavity, the large vortex (primary vortex) forms the flame to provide stable heat source, the small vortex (secondary vortex) prompts the rapid mixing of the mainstream and high temperature fired gas in the cavity, which timely ignites fresh mixture to further improve combustion efficiency. But when mixture and gas are injected from the forewall and afterwall, how to determine their injection position and velocity matching becomes a very critical issue. It is very inconvenient in practical applications. Agarwal [14,15] combined a flow guide vane at the leading edge of the cavity with TVC, part of the mainstream is diverted into the cavity, which can easily form a double vortex structure stability, but combustion efficiency is not high.

So, a blunt body attempts to be arranged in downstream of TVC with flow guide vane. On one hand, it can strengthen the mixing of mainstream and cavity gas, on the other hand, reverse flow zone can be formed to make mixture combust sufficiently behind the blunt body. In this paper, numerical simulation of cold state and combustion state unsteady turbulence flow in three dimensional TVC with or without blunt body was performed, effect of the blunt body on flow was analyzed.

2. Geometric and condition

2.1. Geometric of model

Fig. 1 shows the geometric model of three dimensional TVC with a L-shape flow guide vane and blunt body. $L = 250$ mm is defined as the length of the combustor, $L_i = 50$ mm is defined as the depth of the combustor inlet or exit, the flow guide vane has a thickness of 1 mm. The size of the inlet face is 50 mm × 30 mm. $H_f = 30$ mm is defined as the depth of the cavity, $L = 36$ mm is defined as the length of the cavity. Position parameters (a, b, c) of the flow guide vane and structure parameters (S, L2, L1) of the blunt body are marked in Fig. 1. In this paper, a is defined as the length of guide vane inside the cavity, b is defined as the distance from guide vane to the cavity leading edge, c is the distance from guide vane to the cavity forewall. $a/H_f = 0.5$, $b/L_i = 0.2$, $c/L = 0.1$. S is the distance from the blunt body to the cavity forewall, L2 is defined as the length of the blunt body, L1 is the height of the blunt body. $S/L = 0.7$, $\mathrm{L2}/L = 0.1$, $\mathrm{L1}/L_i = 0.25$.

2.2. Boundary conditions

Numerical simulation is performed by incompressible Navier–Stokes equations and realizable k-ε model [16] is adopted as the turbulence model, time step is set to 1.0×10^{-5} s. The method of pressure and velocity coupling is SIMPLE [17], the second-order central difference scheme is used for the diffusion terms, and the

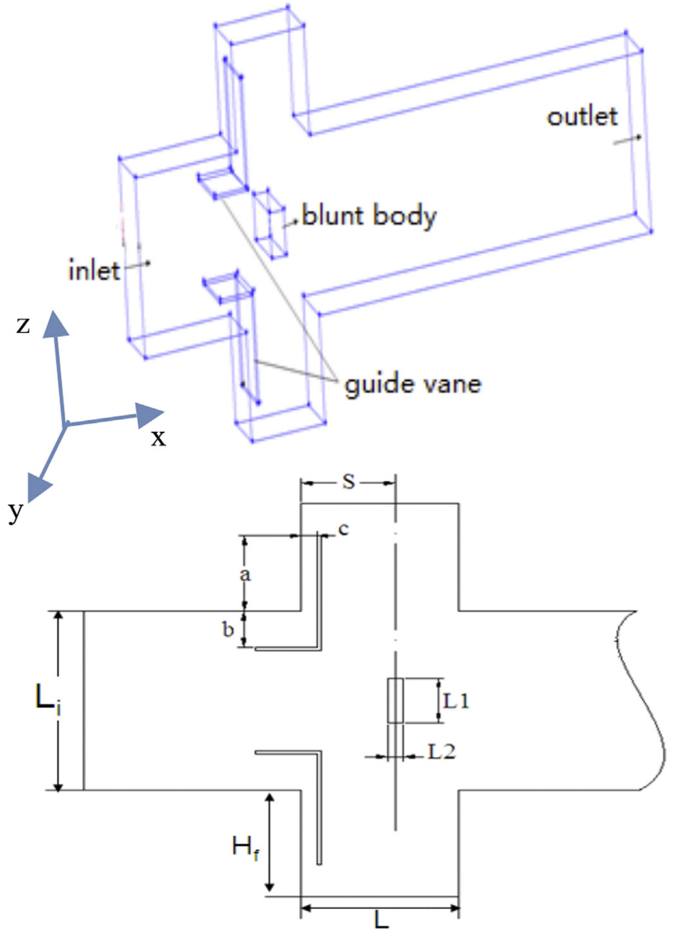


Fig. 1. The geometric model of TVC.