Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 141 (2017) 287–291

Energy Procedia

www.elsevier.com/locate/procedia

4th International Conference on Power and Energy Systems Engineering, CPESE 2017, 25-29 September 2017, Berlin, Germany

# Numerical investigation of typical valveless pulse combustor working process

Zhang Tao[a], Zhan Yong[b], Cai Wenxiang[b]*

[a]Beihang University, Beijing 100083, P R China
[b]Nanjing University of Science and Technology, Nanjing 210094, P R China

**Abstract**

Due to the valveless pulse combustor's simplicity in structure and reliability in performance, it may be the most ideal low cost high efficient combustion system for agriclutural energy. In a typical valveless pulse combustion, a numerical simulation is performed to understand the variation of flow field and to research the performance of the valveless pulse combustor by use of general CFD simulation software (CFX). Results of numerical simulation indicate that the pulse combustor works felicitously and working process of the pulse combustor is consistent with the basic theory of the pulse combustion.


Peer-review under responsibility of the scientific committee of the 4th International Conference on Power and Energy Systems Engineering.

*Keywords:* valveless pulse jet, numerical simulation, multi-cycle component, pulse combsution;

## 1. Introduction

With the rapid development of the new concepts combustion system of agricultural resources, there's a urgent need for an efficient, affordable and durable small scale propel engine [1-3]. Recently, a lot of research attention has been paid to research the performance and the working process of the valveless pulse combustor. Within the past two decades, pulse combustor has gained ebullient research attention for its great heat capacity intensity, air-breathing, simplicity, less moving parts and extremely low cost [4].

It's very difficult to observe transient flow field in the valveless pulse combustor's working process for the adverse work circumstance. Absence of accurate effective measurement methods and drastic dynamic change in the

* Corresponding author. Tel.: +086-025-84315456
*E-mail address:* haidaicai@sina.com

1876-6102 
Peer-review under responsibility of the scientific committee of the 4th International Conference on Power and Energy Systems Engineering.
10.1016/j.egypro.2017.11.107

working process of pulse combustor cause that experiment research of the pulse combustor is very expensive and difficult for the optimization design of the pulse combustor. With the development of the computational fluid dynamics (CFD), numerical simulation is becoming a useable method to understand combustion process and phenomena [5-7]. Numerical research can be used to observe detailed internal parameters of the pulse combustion, which is very hard or extremely expensive if just gained by experiment research to provide help in the optimization of the pulse combustor.

An estimation model was established to predict the valveless pulse combustor's performance in Ref.8, which is based on the thermodynamic theory and acoustic theory. This estimation model can't supply the information of the three-dimension turbulent reaction flow field, which is the key influence factor in the working process of the pulse combustor. In this paper, a numerical simulation is conducted to understand how valveless pulse combustor works and to study multi-cycle working process in this valveless pulse combustor by using of general CFD package (CFX$^{TM}$ 14.0).

## 2. Numerical models and methods

The computations are performed on IBM Core2 Duo 2.26GHz processor. Typical simulation time for whole simulation (0.06s) is about 120h. In numerical research, the second order transient scheme and high-resolution advection scheme and high-resolution advection scheme are used to discrete general governing equation. The time step is set to $2\times10^{-5}$s and the convergence criterion is set to $10^{-4}$ in residual mean square value.

### *2.1. Grid of valveless pulse combustor*

A typical valveless pulse combustor, as shown in Fig.1, is simulated numerically to understand the change process of pulse combustion flow field. Diameter of the intake is 22mm, diameter of the combustor is 62mm and diameter of the tail is 32mm. Other key dimensions can be found in Fig.1. Total number of the grid is 88082, which are generated by using of software GaimbitTM 2.0. Fig.2 shows the detail of the intake and the combustor. Fig.3 shows the grids of the tail. The grids of the intake and the tail belong to orthogonal hexahedral mesh. The grids of the combustor belong to the tetrahedral mesh for considering the detailed structure of the fuel inlet in the combustor.

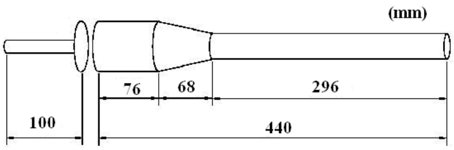


Fig.1 structure diagram of china-type pulse combustor

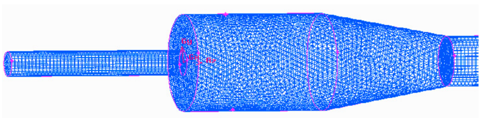

Fig.2 mesh of researched pulse combustor

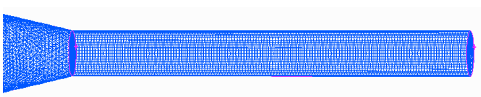

Fig.3 mesh of whole flow field

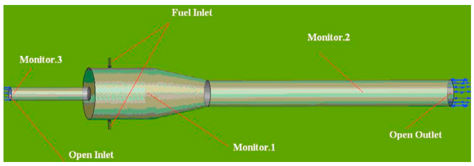


Fig.4 boundaries and monitor points

### *2.2. Boundary condition*

In numerical simulations, the propane is used as the fuel, which is fed into the pulse combustors through the fuel inlet at a constant flow rate of 0.4g/s. In order to consider the exchange processes of mass, momentum, energy and components between the intake, combustor and tail, connected planes of different zones are dealt as the interface boundary. As shown in Fig.4, three monitoring points are set to observe the variation process of the special parameters.