Available online at www.sciencedirect.com

ScienceDirect

Procedia Technology 24 (2016) 420 – 427

Procedia Technology

International Conference on Emerging Trends in Engineering, Science and Technology (ICETEST - 2015)

Flow Field Predictions of Bluff Body Introduced Micro combustor

Sarath C.G.[a],Sreejith M.[b],Reji R.V.[c]

[ab]*Department of Mechanical EngineeringLBS College of Engineering Kasaragod 671542*
[c]*Department of Mechanical Engineering College of Engineering Trivandrum 695016*

Abstract

Combustion in micro scale is the area having wide scope in this era of miniaturization of energy efficient devices. Some important areas of application are in the field of propulsion as a micro thruster and as an apt replacement for conventional batteries by accounting the high energy density of micro combustors. Numerical simulation of combustion is a challenging process due to associated chemical reaction, unsteadiness and non equilibrium nature. In this work a numerical study has been carried out in a typical non premixed micro combustor. The steady simulations of micro combustor without and with bluff body are analyzed for different fuel inlet velocities. For studying the effect of location of bluff body, the micro combustor with wall positioned bluff body is separately analyzed by changing the position of bluff body along axial direction. The effect of different bluff body shapes and significance on its location in combustion process is analyzed. The results are presented in the form of contours of velocity, temperature and mass fractions.
*Keywords:*Micro combustion,Methane air reaction,Bluff body,Premixed combustion

1. Introduction

In order to extract maximum energy from available fuel, miniaturization of devices is preferable because they offer better control, more precise measurements and more flexibility than the conventional ones. However, the newer applications also demand greater sophistication in efficiency, weight and reliability; for example in space propulsion and battery operated devices. The energy needed for operating these devices is obtained by using a number of methods including batteries, fuel cells and most importantly, combustion. Combustion is the most important route to convert chemical energy of various fuels like hydrogen, natural gas, fuel oils, gasoline and alcohols into thermal energy. Due to obvious restriction on available volume in miniature devices, combustion based energy converters need to operate at necessarily small length scale in these devices. In this respect, combustion is termed as micro-combustion because the characteristic length scale of the confined space is of the order of 100-1000

[*]**Sarath C G** Mob.NO: 8907432281 *E-mail aaddress*: **sarathcg06@gmail.com**

Peer-review under responsibility of the organizing committee of ICETEST – 2015
doi:10.1016/j.protcy.2016.05.058

μm.[1] If it is possible to integrate such a "micro-combustor" in miniature devices, then its attributes such as flexibility in terms of distributed and autonomous operation, enhanced overall efficiency, reduction in emissions and improvement in economy and reliability become visible and useful[7]. The present work is an attempt to learn how the introduction bluff body in the flow field of micro combustor affects its performance characteristics. For getting more clarity in the result different bluff body combinations are analysed. The study is also extended to the positioning of wall positioned bluff body in the micro combustor. The numerical study is carried out using ANSYS Fluent 15. The micro-scale flows are generally characterized by non equilibrium wall condition and difficulty in calculating the wall friction velocity [2].

By accounting physical measurement of combustor, the characteristics length of the combustor is more than the molecular mean free path (λ) the continuum approach and standard conservation equations are valid[3]. But most of the published works reports that the equilibrium/continuum approach is valid for their cases of consideration [3][4]. For species transport combustion model along with k-e turbulence model is adopted for this study [4]. The main issue associated with micro combustor is the instability in flame due to low residence time so the major portion of the charge is leaving the combustor as unburned. The presence of recirculation zone can minimize this scenario to some extend [5]. Being small in size it is very difficult to measure the parameters inside the micro combustor and more than that more sophisticated measuring devices are required. So an effective method of analyzing flow characteristics is numerical analysis.

The main aim of this work is to predict the performance improvement in micro combustor by incorporating bluff bodies of different shape. Its performance parameters are analysed for different fuel inlet velocities by keeping a fixed oxidiser velocity. The experiment is repeated by changing the position of bluff body, the performance parameters such as temperature, velocity, rate of combustion were analysed using contours and plots

Nomenclature

u	Component of velocity
$\bar{u}$	Steady component of velocity
u'	Fluctuating component of velocity
ρ	Density
μ	Kinematic viscosity
δ_{ij}	Kronecker delta
A_r	Consistent pre exponential factor
T	Absolute temperature
K	Turbulence kinetic energy
P	Pressure
K_{eff}	Effective conductivity

2. Numerical modelling

Mathematical modeling of the problem is done by Computational Fluid Dynamics (CFD), a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. In CFD, the conservation of mass, momentum, energy is evaluated in the flow domain. Based on the physics associated with the problem continuity, Navier-Stokes equation, Energy equation, ideal gas approximation, Arrhenius equation, turbulence equation, etc. are evaluated in the simulation process. The iteration up to a particular level of convergence selected for proper prediction of results. In solving step the associated equations are solved iteratively as a steady-state or transient. After adequate convergence the results are evaluated in post-processing step. The