Fuel Processing Technology 166 (2017) 258–268

Contents lists available at ScienceDirect

Fuel Processing Technology

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

Research article

Combustion chamber design and performance for micro gas turbine application

Ibrahim I. Enagi [a], K.A. Al-attab [b], Z.A. Zainal [c,*]

[a] Department of Mechanical Engineering, School of Engineering Technology, Federal Polytechnic, P.M.B. 55, Bida, Niger State, Nigeria
[b] Department of Mechanical Engineering, Faculty of Engineering, Sana'a University, Sana'a, Yemen
[c] School of Mechanical Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Penang, Malaysia

ARTICLE INFO

Article history:
Received 23 January 2017
Received in revised form 31 May 2017
Accepted 31 May 2017
Available online xxxx

Keywords:
Micro gas turbine
Turbocharger
Combustion chamber
Flame holder
ANSYS-FLUENT

ABSTRACT

Micro-gas turbines (MGT) are small-scale independent and reliable distributed generation systems that offer potential for saving energy and reducing carbon monoxide (CO) emissions. They are expected to play a vital role in future energy supplies for remote locations with or without grid connections. In this paper, a design and development of a combustion chamber for micro-gas turbine was performed by SOLID-WORKS and computational fluid dynamics (CFD) ANSYS-FLUENT simulation software. Different chamber geometries were used to simulate with species transport and non-premixed combustion models to determine the optimum chamber design. The best chamber geometry adopted after optimization was 50 mm flame holder diameter, 60 cm chamber height, having 4 holes of 6, 8 and10 mm with dead zone between the combustion zone and dilution zone. A two-stage MGT was developed based on vehicular turbochargers to test the chamber. The experimental test of the chamber with liquefied petroleum gas (LPG) fuel resulted in a stable combustion with CO emission below 100 ppm and turbine inlet temperature below 900 °C.

1. Introduction

Micro-gas turbine (MGT) usually produces between 25 and 500 kW of electrical power, it has minimal maintenance and operational cost, high power density and low emission. An important factor that attracts researchers to develop MGT's especially for renewable energy fuel types is that it can be operated with various kinds of fuels [1]. Micro-gas turbine in recent times is given attention for decentralised generation of renewable energy [2]. There has been a renewed interest on the MGT development and deployment on small scale distributed cogeneration (DG) and poly-generation concepts [3]. When compared with other technologies for small scale power generation, they offer numerous advantages which include, high grade heat and lower emission levels, compact size, reduced noise and vibrations, easy operations and installations [4].

MGTs are essentially of two types, the first is composed of a high speed single shaft unit carrying centrifugal compressor and radial turbine (ranges from 50,000 RPM to 120,000 RPM) on the same shaft as an electrical synchronous machine. In this design, the compressor speed remains constant at generator rated speed resulting in a significant drop in efficiency at part load [5]. The second type uses a power turbine rotating at 3000 RPM connected to a conventional generator via gear box in a split- shaft designed whereas the compressor speed varies with output load resulting in a better part load efficiency [6,7].

The developmental work in small stationary and automotive gas-turbines were initiated by automotive industries in 1950′s and served as fundamental achievement in today's MGT technology [8]. Modern MGT combines the reliability of an aircraft generator with low cost automotive turbocharger [8]. The common technique used to increase the power density of internal combustion engines is turbocharging which reduces pollutant emission and fuel consumption [9]. The overall turbocharger performance and turbine power have significant effects on the thermal energy transfer and engine volumetric efficiency [10,11].

The design of a combustion chamber based on temperature homogeneity and CO emission is a critical issue in the development of the MGT. Although, some typical design and experimental studies on different micro combustor configurations were carried out in the past, there is need for further design and development to improve on fundamental issues such as low outlet temperature and CO emissions. Flame sustainability over a range of operating mass flows and air-fuel (AF) ratios in a high power density micro combustion were investigated [12,13]. A high and uniform temperature distribution along the wall of the micro combustor flame tube was achieved using a stainless steel based combustor [14,15]. Another micro-axial stainless steel based combustor was developed to improve thermal performances of MGT [16,17].

* Corresponding author.
E-mail address: mezainal@usm.my (Z.A. Zainal).

http://dx.doi.org/10.1016/j.fuproc.2017.05.037
0378-3820/

The applications of simulation software have rapidly increased over the last decade in improving many complex processes including combustion chambers. Computational fluid dynamics (CFD) has been extensively used as an important design tool of combustion chambers. Numerous studies have utilized CFD simulation for the design and optimization of burners [18,19] and combustion chambers in MGT [6,20–22], aviation and industrial gas turbines [17,23].

In the present work, a combustion chamber was designed and developed using CFD ANSYS-FLUENT simulation software. Species transport and non-premixed combustion models were used to determine the optimum flame holder chamber geometries. The designed combustion chamber was investigated through optimization process and the effects of different variables and geometries on the combustion stability were studied. Combustion stability was also verified experimentally during the initial tests of a two-staged turbocharger based MGT.

2. Combustion chamber design

For chamber design, a criterion for selecting suitable combustor geometry was carefully examined followed by designed calculation of the dimensions. The combustion chamber design is faced with inherent challenges such as emission control, chamber materials temperature limitations and flame stability. The steps taken in designing the chamber are: SOLIDWORKS drawing of the chamber, and simulation using ANSYS-FLUENT software.

2.1. SOLIDWORKS drawing of the chamber

The combustion chamber consists of the air jacket and flame tube with 100 mm fixed height conical exit of the chamber. Chamber inlet and outlet diameters were also fixed at 50 mm diameter to match the specifications of the first stage MGT Garrett GT25. Main manipulated geometries were: chamber height, chamber diameter, flame tube diameter and combustion zones configurations on the flame tube. Flame tube diameter was varied in the range of 50–150 mm, thus, chamber diameter was varied accordingly to leave air jacket space of 25 mm, while chamber height was varied from 300 to 1000 mm. Simple flame tube geometry with fixed rows of holes of 6 mm diameter through the tube height was studied first. However, for the final flame tube optimization, three zones were identified on the flame tube: premix zone, primary combustion zone and dilution or cooling zone. Each zone has a number of rows with a certain number of holes on teach row with space between zones. The number of rows was varied from 4 to 15 for each zone and number of holes in each row was also varied from 4 to 10. Two holes diameters arrangements in respective to the zones were studied: 6, 8, 10 mm and 8, 10, 12 mm.

In order to simplify the meshing and simulation processes, flame tube was built as a void inside the chamber instead of a solid body. This was achieved by creating two separate bodies (chamber and flame tube) then subtracting the flame tube body from the chamber using Combine: Subtract feature in SOLIDWORKS.

2.2. Simulation using ANSYS-FLUENT software

Combustion chamber geometry was optimized using ANSYS 16.1 CFD simulations program. In the Work-bench, chamber geometry was imported from SOLIDWORKS to the Design-Modeler tool. Meshing (ANSYS ICEM CFD) tool was used to create the mesh and test its quality. After that, 3D simulation was performed on the mesh using Fluent CFD tool. Species transport combustion model was tested first followed by the non-premixed combustion model. The chamber is aimed at testing liquid and gaseous fuels with MGT, thus, LPG (butane/propane) and diesel ($C_{12}H_{23}$) fuels were tested. For this initial design simulation, no swirlers were used. However, different type of commercial diesel injectors/swirlers will be simulated and compared to the experimental test with liquid fuels in future work. Air inlet boundary conditions were determined based on Garrett GT25 turbocharger specifications and compressor maps. Maximum air flow rate of 0.15 kg/s at 1.4 barg was considered to achieve compressor efficiency of about 70%. Optimum chamber should achieve a complete combustion with low CO emissions and stable flame propagation inside the flame tube in a compact form. Table 1 shows the main boundary conditions for air and fuel inlets, chamber outlet and walls.

2.2.1. Species transport model

For flow analysis, k-epsilon viscosity model was used since it is recommended by the FLUENT theory guide for laminar-turbulent transition and turbulent flow ranges. Reynolds stress model will be used in the future to simulate different type of commercial diesel swirling injectors to be compared with the experimental test with diesel fuel. As for the combustion, energy equations were activated (i.e. non-adiabatic combustion) with species transport model used first due to its lower computing demand compared to the non-premixed combustion model. This step will provide the initial estimation of the minimum size in which the combustion will start taking place. Diesel-air and propane-air mixture were tested separately at 20% excess air with 0.15 kg/s air flow rate. And with such high flow, turbulent combustion is expected in small geometries while larger volumes will experience laminar flows in some parts of the chamber. Therefore, both laminar Finite-Rate and the turbulent Eddy-Dissipation equations were used in the volumetric combustion model. Main geometry parameters studied with this model were the chamber height and flame tube diameter. Other variables such as chamber diameter and flame tube holes configuration were neutralized. For chamber diameter, air jacket width of 25 mm was fixed, thus, making this variable dependent on the flame tube diameter variable. As for the flame tube configuration, 8 holes of 6 mm diameter were used in each row with 10 mm distance between the rows. Thus, number of rows will increase accordingly with the chamber height. Geometries with sustainable combustion were then further optimized using radiation and non-premixed combustion models.

2.2.2. Non-premixed model

To achieve stabilized combustion in MGT combustors, an improved understanding of turbulent combustion through CFD simulation is required. Moreover, the accuracy of this simulation is highly dependent on the turbulence and combustion models. The non-premixed combustion model which employs the infinitely fast chemistry assumptions

Table 1
Parameters set out in boundary conditions.

Parameters	Values
Fuel inlet	
Temperature	300 K
Pressure	2 barg
Mass flow rate (LPG) (20%–70% excess air)	0.0053–0.0075 kg/s
Mass flow rate (Diesel) (20%–70% excess air)	0.0056–0.0078 kg/s
Air inlet	
Temperature	530 K
Pressure	1.4 barg
Mass flow rate	0.15 kg/s
Outlet	
Pressure	1.4 barg
Back flow temperature	600 K
Inner walls	
Materials	Steel
Emissivity	0.5
Outer walls	
Materials	Steel
Wall thickness	6 mm
Heat fluxes	$-10,800$ W/m^2