Combustion and Flame 185 (2017) 117–128

Contents lists available at ScienceDirect

Combustion and Flame

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

# A jet-stirred chamber for turbulent combustion experiments

Abbasali A. Davani*, Paul D. Ronney

*Department of Aerospace and Mechanical Engineering University of Southern California, Los Angeles, CA 90089, United States*

ARTICLE INFO

*Article history:*
Received 8 November 2016
Revised 11 July 2017
Accepted 11 July 2017
Available online 22 July 2017

*Keywords:*
Turbulent combustion
RANS
Reynolds stress
Jet-stirred reactor
Fan-stirred reactor

ABSTRACT

A new type of jet-stirred chamber (JSC) incorporating multiple impinging turbulent jets is proposed as an apparatus for the study of turbulent premixed flames. ANSYS-FLUENT computations using a RANS - Reynolds Stress Model were used to simulate the flows inside the JSC and identify an optimal configuration of inlet jets and outlet ports providing the most nearly ideal flow, i.e. homogeneous and isotropic with large turbulence intensity compared to the mean velocity. A configuration of 8 jets, each surrounded by a concentric annular outlet, at the corners of an imaginary cube circumscribed by a spherical chamber, produced by far the most nearly optimal flow characteristics. The performance of this configuration, called Concentric Inlet And Outlet (CIAO), was compared quantitatively to two popular fan-stirred chamber (FSC) designs and the CIAO JSC was found to provide far more nearly ideal flow properties. The robustness of the CIAO design was demonstrated by intentionally misaligning the jets or mismatching their flow velocities. A comparison of simulated turbulent flames in CIAO and an FSC showed that CIAO enabled far more nearly spherical expanding flames with nearly the same inferred turbulent burning velocity ($S_T$) regardless of the flame radius $r_f$ and the value of the mean progress variable ($\bar{c}$) used to define the flame location, whereas in the FSC the flame was not nearly as spherical and there was considerable variation of $S_T$ depending on $r_f$ and $\bar{c}$.



## 1. Introduction

It is well known that turbulence increases the propagation rates ($S_T$) of premixed-gas flames by wrinkling the flame front and increasing its surface area, thereby leading to higher rates of thermal enthalpy production per unit volume [1,2,3]. For this reason, turbulence is used in essentially all power generation devices employed premixed combustion, e.g., spark ignition internal combustion engines and premixed gas turbines. Many experimental apparatuses have been used to study premixed turbulent combustion including Fan-Stirred Chambers (FSCs) [4,5,6,7], Bunsen flames [8,9,10], slot-jet generated turbulence [11], rod-stabilized V-shaped flames anchored downstream of a jet employing fractal grids to generate turbulence [12] and opposed-jet counterflows [13,14]. Many models of premixed turbulent combustion assume homogeneous and isotropic turbulence over many integral scales of turbulence ($L_I$) with zero (or at least a constant and uniform) mean flow. A persistent issue with the aforementioned apparatuses is how nearly the turbulence they generate conforms to the simplifying assumptions made by these models. In this work we propose a new apparatus that produces more nearly homogeneous and isotropic turbulence with small mean flow, develop a numerical model to simulate its behavior and compare the performance of this apparatus to FSCs.

This proposed apparatus is motivated as follows. It is well known that a pair of counter-flowing turbulent jets produces a nearly constant turbulence intensity ($u'$) along the jet axis [14,15,16,17]. While this provides homogeneity only in the axial coordinate direction, multiple pairs of impinging jets aimed towards a central location could in principle provide nearly homogeneous, isotropic, zero-mean turbulence in all coordinate directions. Of course, the jet inflows necessitate a set of outlet ports if the test section is to be is contained within a combustion chamber; in practice the chamber inlets and outlets would need to be coupled via a recirculating-flow system or operated in a blow-down mode. While jet-stirred chambers (JSCs) are widely used in chemical kinetics studies [18,19] under conditions where the chemical reaction time scale is much longer than the turbulent mixing time scale (*i.e.*, at low Damköhler number, Da) so that the reactants and products are presumably well-mixed within the chamber, they are not typically employed for studying propagating turbulent premixed flames (*i.e.*, at high Da where thin, distinct reaction zones exist). Nevertheless, compared to traditional FSCs and other apparatuses, the proposed JSC approach has several potential advantages:

1. Any number and configuration of inlet jets and outlet ports can be used to create a more nearly homogenous, isotropic, zero-mean flow.

* Corresponding author.
*E-mail address:* davanida@usc.edu (A.A. Davani).

http://dx.doi.org/10.1016/j.combustflame.2017.07.009

2. A single pump or blowdown system, external to the combustion chamber, can be used to drive the flow.
3. There are no shafts penetrating the chamber wall that need to be sealed; only static jets and ports breach the chamber wall.
4. There is no flow bias due to swirl created by fans.
5. Any desired amount of swirl can be deliberately introduced in a well-controlled manner.
6. Since there are no moving parts, complicated geometries with multiple sets of jets and ports are readily designed using CAD software and constructed using 3D printing techniques.

In addition, JSCs retain many of the advantages of FSCs over Bunsen flames, grid-generated turbulence and other apparatuses with an appreciable mean flow, e.g.

1. $u'$ is nearly independent of the mean flow.
2. The walls are remote from the flame, thus conductive heat losses are negligible.
3. The flames are freely propagating as opposed to conforming their location and shape in response to being stabilized in the presence of a mean flow field.
4. $S_T$ is directly inferred from the expansion rate of the quasi-spherical flame front.
5. The flames are not subject to a mean strain (as in a counter-flow) or mean shear.
6. The effects of ambient pressure are readily assessed.

Consequently, the objectives of the current study are to (1) examine computationally the flow properties of a variety of JSCs using multiple impinging jet configurations; (2) determine the JSC configuration that provides most nearly homogeneous and isotropic turbulence with large turbulence intensity compared to the mean velocity; (3) determine how imperfections in the manufacturing and operation of the preferred JSC affect its performance and (4) compare this JSC to other common turbulent combustion apparatuses, in particular FSCs, with respect to both cold-flow properties and simulated propagating turbulent flames.

## 2. Numerical model

Simulations were performed using the Reynolds Averaged Navier Stokes (RANS) approach with the Reynolds Stress Model (RSM) for turbulent transport. In the RSM, transport equations (with source and sink terms) for the turbulent kinetic energy dissipation rate $\varepsilon$ and each of the six independent Reynolds Stress terms are solved, resulting in seven equations for the turbulence properties in 3D simulations. These turbulence properties provide the required turbulent transport rates for use in the momentum and energy conservation equations. Although RSM is a second order closure method and is generally considered less accurate than more advanced techniques such as Large Eddy Simulation (LES) or Direct Numerical Simulation, RSM requires far less computational effort. For this work ANSYS-FLUENT computational fluid dynamics software version 17.0 was used to solve the RANS-RSM equations. By comparison with experiments it will be shown that this RANS-RSM/FLUENT approach likely provides sufficient accuracy for the class of problems investigated in this work. The Pressure-Implicit with Splitting of Operators (PISO) scheme with second order upwind spatial discretization and first order implicit time discretization for transient simulations was used to solve the Navier–Stokes equations. Grid independence was confirmed by comparing results with different grid resolutions. Between $2 \times 10^6$ and $5 \times 10^6$ volume elements were used for the various JSCs and FSCs modeled in this work. The steady-state flow solver was employed where applicable and spot-checked by using the results of converged steady solutions as starting conditions for transient simulations. In such cases the steady solutions did not diverge; this indicates a measure of stability of the results obtained although it does not represent a formal stability analysis. For FSCs and for the JSCs with combustion, the phenomena of interest are inherently unsteady and in these cases fully transient simulations were conducted.

A relevant test of the viability of our simulations for the current purposes is the flow between a pair or multiple pairs of impinging turbulent jets. No prior studies of multiple pairs of impinging jets could be found, and Pettit et al. [20] provides the only well-characterized experimental study of a single pair of impinging turbulent jets known to us. The reported properties Pettit et al.'s experiment are as follows: jet and co-flow mean outlet velocity 6.58 m/s, turbulence intensity 1.97 m/s, jet inner diameter 12.7 mm, jet wall thickness 1.65 mm, annular co-flow outer diameter 29.5 mm and jet spacing 19 mm. All these features were incorporated into the simulations. The stated accuracy of Petit et al.'s Particle Image Velocimetry (PIV) measurements of turbulent intensity was better than ±10%. Figure 1 shows a comparison of the RSM simulations with the experiments and LES predictions of Pettit et al. All the key features of the flow are captured well by the RSM simulations (in fact, as well as the LES modeling), both with regards to the differences between the axial ($u'$) and radial ($v'$) velocity fluctuation characteristics and their magnitudes. In particular, note that the experiments and simulations all show that $u'(z)$ (where $z$ is the axial coordinate) is inhomogeneous, reaching a peak at the stagnation plane ($z = 0$), whereas $v'(z)$ is nearly homogenous and of smaller magnitude than $u'$. Mean-flow properties (not shown) exhibited similar agreement.

## 3. Polyhedral jet stirred reactors

With this initial test of the viability of the numerical model providing a level confidence in its predictive capability, five spherical chambers of 30 cm diameter with the inlet jets and outlet ports flush with the surface of the sphere arranged in accordance with the geometries of the faces of Platonic solids were simulated (Fig. 2, top [21] and second rows). For each geometry, each inlet air jet diameter is 2 cm and the annular outlet diameters are chosen to obtain the same total outlet area as inlet area. For these 5 configurations, to make a "fair" comparison the total volume inflow rate is held fixed at 0.0502 $m^3/s$, corresponding to a mean jet exit velocity of 40 m/s and a Reynolds number of $5.33 \times 10^4$ for the 4-jet tetrahedral case with proportionally lower values for the cases with more inlet jets. In all cases the turbulence intensity at the jet exits was fixed at 20% of the mean jet exit velocity. Inlet boundary conditions were plug-flow with the specified inlet velocity and turbulence intensity and 1 atm ambient pressure outflow conditions were prescribed at all outlets.

Figure 2 shows that as the number of jets is increased, the homogeneity of the turbulence intensity (third row) appears to increase and the magnitude of the mean flow (bottom row) relative to the turbulence intensity appears to decrease, but *quantitative* measures of these properties is needed. Toward identifying quantitative measures, first recall that for RANS modeling the flow velocity at any location $i$ (i.e., for any volume element in this finite-volume simulation) is divided into time-averaged mean velocity components $\overline{u_{x,i}}$, $\overline{u_{y,i}}$ and $\overline{u_{z,i}}$ and time-averaged fluctuating components $u'_{x,i}$,$u'_{y,i}$ and $u'_{z,i}$. From these components, we can define the mean velocity magnitude $\overline{u_i}$ and turbulence intensity $u'_i$ at location $i$ as follows:

$$\overline{u_i} \equiv \sqrt{\left(\overline{u_{x,i}}\right)^2 + \left(\overline{u_{y,i}}\right)^2 + \left(\overline{u_{z,i}}\right)^2};$$

$$u'_i \equiv \sqrt{\frac{1}{3}\left[\left(u'_{i,x}\right)^2 + \left(u'_{i,y}\right)^2 + \left(u'_{i,z}\right)^2\right]} \quad (1)$$

For a volume $V$ comprised of $n$ individual cells having volumes $v_i$ we can define a mean turbulence intensity $\overline{u'_V}$ volume-averaged