International Journal of Thermal Sciences 102 (2016) 185–201

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Parametric study of flow field and mixing characteristics of outwardly injected jets into a crossflow in a cylindrical chamber

S.A. Nada [a,*], A. Fouda [b], H.F. Elattar [a]

[a] Mechanical Engineering Department, Faculty of Engineering, Benha University, Benha, 13511 Qalyubia, Egypt
[b] Mechanical Power Engineering Department, Faculty of Engineering, Mansoura University, 35516 El-Mansoura, Egypt

ARTICLE INFO

Article history:
Received 8 April 2015
Received in revised form
23 November 2015
Accepted 23 November 2015
Available online xxx

Keywords:
Parametric study
Multiple rows of jets
Crossflow
Cylindrical chamber
Penetration depth
Mixing characteristics

ABSTRACT

Flow field and mixing characteristics of coolant radial jets injected radially from multiple nozzles rows of centreline distributer into a heated non-reacting crossflow in a cylindrical chamber is numerically investigated. The study simulates the process of air jets injection in the combustion chambers of gas turbines through discrete holes. Three-dimensional model using ANSYS-FLUENT 14.5 CFD package is used. The effects of jet-mixing ratio, nozzles diameter, distributer diameter, number of nozzles rows, number of nozzles per row on the penetration depth and mixing quality through chamber cross section are parametrically studied. The results are validated with the available experimental data and good agreement are obtained. The results showed that nozzle diameter, distributer diameter and jet-mixing ratio have effects on the penetration depth and the mixing quality compared to the effects of the number of nozzles per row and the number of nozzles rows. The effects are remarkable at high mixing ratio. The integral mixing quality along the chamber cross section increased with increasing mixing ratio, nozzles diameters, number of nozzles per row and number of nozzles rows. Dimensionless correlations for predicting the penetration depth and mixing quality in terms of the controlling parameters are developed.

1. Introduction

Nowadays jet mixers are considered as one of the common devices of mixing. Jet mixers injected normally into crossflow are used in many industrial and engineering applications because their high transfer coefficients and mixing efficiency with relatively low pressure drops. Primary applications of jet-in-crossflow (JIC) include chimney plumes, discharges into lakes and streams, and vertical and/or short take-off and landing (V/STOL) aircraft aerodynamics. Additional applications are chemistry, primary combustions, burners industries, industrial mixing, cooling of electronic packages, tempering of glass and metal process and cooling of gas turbine surfaces and mixing of air and fuel/combustion gases in gas turbine engines.

These widespread applications of jet mixers have led over the past 60 years to experimental, theoretical and numerical examinations of the flow field associated with jets in crossflow. One of the most important applications of jets in crossflow is in gas turbine engines where the heat release in turbine engine is mainly controlled by mixing process between air and fuel/combustion products, so it is important to enhance the mixing process for achieving the optimum performance of turbine engine. The air is injected into the dilution zone in gas-turbine engines where the jets are injected radially into the combustion chamber through discrete holes along its circumference. The targets of these injections are to stabilize the process of combustion and to dilute the hot combustion products. The objective of many researches has been modelled the penetration depth and mixing characteristics of coolant air jets injected into a confined heated crossflow. Extensive reviews on jets in cross flow are available in Margason [1], Demuren [2], and Karagozian [3]. These reviews revealed that the use of simulation tools based on computational fluid dynamics (CFD) could provide a strong asset for the development of jet mixer in cross flow. Ramsey and Goldstein [4] performed an experimental study on the mixing process of a heated jet injected a deflecting flow into a crossflow in a wind tunnel. The jet flow was introduced at different angles to the mainstream flow directions. Temperature profiles in the interaction region and velocity and turbulence-intensity profiles are reported for different

* Corresponding author. Tel.: +20 1066611381 (mobile).
E-mail address: samehnadar@yahoo.com (S.A. Nada).

http://dx.doi.org/10.1016/j.ijthermalsci.2015.11.017
1290-0729/

Nomenclature

C	π *sqrt(2 J)/*$N_{n,r}$; derived in Ref. [12]
D_d	distributer diameter, m
D_m	mixing chamber diameter, m
d_n	nozzle diameter, m
J	jet-to-mainstream momentum flux ratio $J = \rho_j V_j^2 / \rho_\infty V_\infty^2$
JIC	jet-in-crossflow
h	penetration depth, m
ID	inner diameter-side
L	mixing chamber length, m
$\dot{m}_j$	jet mass flow rate, kg/s
m_r	jet-mixing ratio (mass basis), $m_r = \dot{m}_j/\dot{m}_t$
$\dot{m}_t$	total mass flow rate, kg/s
N_r	number of nozzles rows
$N_{n,r}$	number of nozzles per row
OD	outer diameter-side
P	fluid static pressure, Pa
Pr_t	turbulent Prandtl number, $Pr_t = \mu_t c_p/k_t = 0.85$
R	Local radial distance, m
R	gas constant, J/kg k
R_d	distributer radius $(D_d/2)$, m
R_m	mixing chamber radius $(D_m/2)$, m
S	Spacing between adjacent jets, m
S_x	Spacing between jet rows, m
T	local temperature, °C
T_j	jet temperature, °C
T_∞	mainstream temperature, °C
V_j	jet velocity, m/s
V_∞	mainstream velocity, m/s
X	local longitudinal distance, m
θ	mixing quality, $\theta = (T_\infty - T)/(T_\infty - T_j)$
$\overline{\theta}$	chamber's exit integral mixing quality; $\overline{\theta} = \sum_{i=0}^{i=n-1} \frac{\Delta R_i}{R_m} \frac{\theta(R_i)+\theta(R_{i+1})}{2}$, $\Delta R_i = R_{i+1} - R_i$
λ	fluid thermal conductivity, W/m.K
μ	fluid viscosity, kg/m.s
μ_t	turbulent viscosity, kg/m.s
ρ	fluid density, kg/m^3
ρ_j	jet density, kg/m^3
ρ_∞	mainstream density

blowing rates. Cox [5] and Holdeman and Walker [6] presented analytical and empirical models for predicting the temperature distribution downstream a row of coolant jets injected normally into a heated crossflow. The effect of controlling geometric and flow parameters are examined. Patankar et al. [7] performed a numerical investigation using a finite-difference solution for the three-dimensional flow field generated by a round turbulent jet deflected by a main stream normal to the jet axis. Results were presented for the cases in which the ratio of the jet velocity to the main-stream velocity ranges from 2 to 10. Wittig et al. [8] studied the development of temperature profile in turbulent mixing of coolant jets with a hot crossflow. Geometry and momentum flux ratios were the dominant parameters. Correlations were presented for opposite wall injection with jets of different momentum flux ratios. Holdeman et al. [9–14] carried out numerous experimental and numerical studies for jet in crossflow to develop, understand and investigate the flow field characteristics resulting from jet mixing. In these studies measures of centre plane parameters and jet shape for unconfined single and multiple jets were conducted. Holdeman [12] presented experimental and computational studies for mixing of single, double and opposed rows of multiple jets inwardly (OD) and outwardly (ID) injections with a confined subsonic crossflow for simulating the dilution process in conventional gas turbine combustors. The results proved that the rate of mixing and penetration of jets into the mainstream were influenced commonly by geometrical parameters and flow variables. Holdeman et al. [13] summarized both experimental and numerical results for the mixing process of multiple coolant jets with a confined crossflow in a circular duct. Holdeman et al. [14] presented experimental and numerical results on the mixing for confined subsonic crossflow in rectangular ducts using opposed rows of jets. Where, ID injection in an annulus each jet penetrates into a sector of increasing width, whereas those in a rectangular duct will mix into a constant-width sector and those injected inwardly, OD from the perimeter of a cylindrical duct will mix into a pie sector. The references [12–14] are extensive and these include all the NASA JIC papers that preceded these summaries. Kroll et al. [15] conducted detailed experimental investigation for optimization of orifice geometry for crossflow mixing in a cylindrical duct. The results showed that the optimized mixing occurs at asymptotic mean jet trajectories in the range of 0.35–0.5. Combined experimental and numerical studies have been carried out by Ahmed et al. [16] and Hart et al. [17] to investigate and improve the understanding of the flow fields of jet development inside a rectangular slot-burners of boilers. The results showed that cross-flow significantly influenced the near field flow development from the slot-burner by deviating both primary and secondary jets from their geometric axes towards the wall. Coletti et al. [18] conducted experimentally investigation of a turbulent jet in cross flow relevant to film cooling applications. The jet is inclined at 30°, and its mean velocity is the same as the cross flow. Magnetic resonance imaging (MRI) and Particle Image Velocimetry (PIV) were used to obtain mean velocity, concentration and Reynolds stresses. The study reported that the counter-rotating vortex pair is critical for the mixing process, the entrainment is weaker than in a transverse jet in cross flow and the constant turbulent Schmidt number model is not adequate for the present flow. Yasaswy et al. [19] carried out experimental investigation to study the local distribution of heat transfer coefficients due to the impingement of a round air jet on a flat plate through a crossflow. It is observed that the distance of the stagnation point from the geometric impingement point and its magnitude increase with the increase in ratio of crossflow velocity to the jet flow velocity. Habibi et al. [20] studied experimentally and numerically the thermal field of a confined low speed slot jet impinging on a hot plate. Galeazzo et al. [21] presented Computational modelling of turbulent mixing in a jet in crossflow. High-resolution measurements using Particle Image Velocimetry combined with Laser Induced Fluorescence have been conducted and used to validate simulations ranging from simple steady-state Reynolds-Averaged Navier Stokes (RANS) to sophisticated Large Eddy Simulation (LES).

Due to the high computational cost, Direct Numerical Simulation (DNS) have been largely restricted to solving turbulent flows in simple configurations [22]. Prediction of engineering flows relies, predominantly, on RANS. There is an increased interest in applying DNS and LES to solve flows in complex engineering geometries. This is a result of advances in parallel computing and affordability of