International Journal of Thermal Sciences 107 (2016) 184–195

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Ceramic panel heating under impinging methane-air premixed flame jets

M. Jarray, K. Chetehouna*, N. Gascoin, F. Bey

INSA Centre Val de Loire – Campus de Bourges, 88 boulevard Lahitolle, CS 60013 – 18022 Bourges, France

A R T I C L E I N F O

Article history:
Received 30 July 2015
Received in revised form
8 April 2016
Accepted 14 April 2016
Available online 22 April 2016

Keywords:
Fireproof burner
Numerical modeling
Impinging flame jet
Heat-transfer
Fluid flow
CFD

A B S T R A C T

Due to the ever wider use of composite materials within aerospace applications, fireproof tests get recently an increased attention. Numerical simulation is expected in the coming years to accompany engineers in their design work to increase the chance of success in the fireproof certification tests. The current research focuses on the numerical investigation of a premixed methane-air flame impinging normal to a flat composite panel. The effects of the exit burner geometry, of the Reynolds number (jet speed) and of the distance between the nozzle and the plate have been investigated. The accuracy and suitability of different turbulence models are discussed. The numerical results are validated with available experimental data. CFD calculations reproduce within 5% the so-called heat transfer efficiency where the realizable k-ε turbulence model demonstrates to be the best. The agreement to the experimental data is maximum (in the following order of importance): i) near the centre of the jet impingement, ii) for higher Reynolds number, iii) for higher distance between the panel and the flame. The Reynolds number increase conducts to an increase of the total heat transfer between the flame and the panel. This is related to the Nusselt number which presents higher value (over 20) in the regions for which the predictiveness of the calculation is found to be better. Efficient modeling parameters are found to reproduce an experimental flame that will serve later in fireproof test simulations.

1. Introduction

Heat transfer due to flame jet impingement is a very important process in industry and it is used for many applications like melting scrap metal, shaping glass, brazing, welding, etc. A lot of research has been carried out in this area, both experimentally and numerically. Most of existing studies on impingement flame jets were concentrated on circular jet utilizing methane or natural gas [1–7]. Examples of experiments quantifying the heat-transfer characteristics of impinging flame jets can be found in references [8–12]. One of the applications that encounter emerging interest and that could take a strong benefit from numerical simulation is related to fireproof certification tests which are imposed by Federal Aviation Agency (FAA) and its European equivalent (EASA). Indeed, the composite panels that form the current planes must be fireproofed and experimental tests show rapid cost increase. Thus, numerical simulation could be a way to investigate panel behavior under fire conditions in order to facilitate related engineering work. To do so, many numerical parameters must be carefully managed in order to ensure the reliability of the calculations.

According to the studies of Dong et al. [13] the premixed butane-air laminar flame jet impingement heat transfer was dependent on Reynolds number, on equivalence ratio of the air/fuel jet, and on configurations of the air-fuel nozzle and of the impingement plate. Chander and Ray [14] presented a very comprehensive and informative review on impingement heat transfer, where flame shape, stabilization and burner geometries are considered. Kwok et al. [15] suggested the importance in matching the flame length with the nozzle-to-plate distance to achieve the best heat transfer performance.

Hsieh and Lin [16] have explored the stability of a methane flame jet impinging normally to a wall. They stated that the flame is relatively stable at lower inlet velocity or lower burner to plate distance.

Hindasageri et al. [17] studied heat flux distribution for premixed methane-air flame jet for Reynolds number varying from 600 to 1400 at an equivalence ratio of 1 for nozzle tip-target plate distance varying from 2 to 4. The temperature distribution of the quartz plate is recorded using a technique based on infrared

---

* Corresponding author.
E-mail address: khaled.chetehouna@insa-cvl.fr (K. Chetehouna).

http://dx.doi.org/10.1016/j.ijthermalsci.2016.04.014
1290-0729/

Nomenclature

A	Area, (m^2)
$C_{2\varepsilon}$,	$C_{1\varepsilon}$ k-ε model constants
g	Gravitational acceleration, (m s^{-2})
G_k	Production of kinetic energy
K	Thermal conductivity, (W m^{-1} K^{-1})
k	Turbulence kinetic energy, (J kg^{-1})
Nu	Nusselt number
P	Pressure, (Pa)
q″	Heat flux, (W m^{-2})
Re	Reynolds number
Su	Burning velocity (m/s)
t	Time, (s)
T	Temperature, (K)
T_w	Temperature at the wall, (K)
T_f	Temperature of the flame (K)
u	Velocity of the fluid, (m s^{-1})
X	Mole fraction
Y	Mass fraction
z	Burner tip to target plate distance, (m)
Z	quartz plate thickness/depth, (m)

Greek letters

α	Thermal diffusivity, (m s^{-2})
β	Radial velocity gradient, (s^{-1})
γ	Heat capacity ratio
Φ	Equivalence ratio
η	Effectiveness
ε	Energy Dissipation Rate, (J kg^{-1} s^{-1})
δ_{ij}	Kronecker symbol
ρ	Fluid density, (kg m^{-3})
σ_k	Turbulent Prandtl numbers for k
σ_ε	Turbulent Prandtl numbers for ε
μ	Dynamic viscosity, ($N\,s/m^2$)
μ_t	Eddy viscosity, (m^2 s^{-1})
v	Kinematic viscosity, (m^2/s)
τ_w	Wall shear stress, (Pa)
$\overline{\Omega_{ij}}$	Mean rate of rotation tensor (s^{-1})
ω_k	Angular velocity, (m s^{-1})

Subscripts/superscripts

aw	adiabatic wall
f	flame
FJ	flame jet
i	initial
j	component of the mixture
m	mixture
w	wall

thermography where the back side and flame side heat flux are compared. These authors stated that the back side measurement would enable capturing thermal images of the impingement plate without loss of aspect ratio, which is the case for flame side measurement, due to obstruction of image by the burner tube. It should be noted that this obstruction can be avoided using numerical approaches.

The aim of this work is to explore the possibility of predicting, numerically, the thermal behavior of a test panel, impinged by a methane-air premixed flame jet. The results of the calculations have been compared to the experimental data of Hindasageri et al. [17].

2. Material and methods

2.1. Experimental data of Hindasageri et al.

Hindasageri et al. reported the spatial distribution of heat flux for 1.1 mm, 3 mm and 5 mm thick quartz plate, and for a varying Reynolds number, nozzle-burner tip distance and equivalence ratio, for square, circular and rectangular burners. The radial distribution of Nusselt number (Nu) and of effectiveness (η) is presented. Perfect circular symmetric heat flux distribution patterns were observed by Hindasageri et al. [17] for circular flame jet issuing from the burners of diameter 10 mm and 8.7 mm. For the burner of diameter 10 mm at $Re = 1000$, the heat flux decreases with the increase in z/d from 2 to 4. For further increase in z/d from 4 to 6, the change in heat flux is negligible. These outcomes are in agreement with the observations reported by Grinstein and De Vore [18] and Miller et al. [19].

2.2. Physical and numerical modeling

The ANSYS Fluent 14 code [20] was used to solve the compressible Navier—Stokes equations in transient conditions. The pressure—velocity coupling was done using the SIMPLE scheme.

2.2.1. Meshing

The quality of a CFD solution is highly dependent on the quality of the mesh. Therefore, it is necessary that the grid is of high quality before proceeding to the next step. The grid structure must be fine enough where strong gradients of the variables are expected. In our study a non-uniform meshes is used with hexahedral and quadrilateral elements, more nodes accumulated around the reaction zone. A grid independence analysis was conducted using five meshes of varying cell number. Each mesh was processed using the enhanced wall treatment, with same boundary condition and a convergence of residual error set to 10^{-4}. The mesh of 1,24,248 nodes and 6,54,742 elements was generated and adopted for circular burner and a mesh of 1,22,295 nodes 6,44,527 elements for square burner (Fig. 1). By examination of different cell sizes of this mesh, no further significant change was found for finer cells; this suggests that the grid independence has been achieved. An example of test configuration is provided for illustrative purpose (Fig. 2).

2.2.2. Computational domain and conditions

The boundary conditions are as follows. At the inlet, a fixed velocity profile is used. The wall boundary is set as no-slip condition wall by default. At the bottom part, the burnt gases flow away through pressure-outlet boundaries. The adiabatic flame temperature is taken at 2200 K and the ambient temperature at 300 K. The 3 mm thick quartz is used.

The impingement plate is made of quartz whose size is 150 mm × 150 mm. The emissivity of the quartz plate reported in the literature is 0.93 [21]. The thermal conductivity k and thermal diffusivity α of the quartz plate is computed at varying temperatures (T(k)) as per Eqs. (1) and (2)