Applied Thermal Engineering 67 (2014) 201–213

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Numerical modeling of flow through an industrial burner orifice

L.C.B.S. Reis [a,b,c], J.A. Carvalho Jr. [a,*], M.A.R. Nascimento [d], L.O. Rodrigues [d], F.L.G. Dias [d], P.M. Sobrinho [a]

[a] *Universidade Estadual Paulista, Campus de Guaratinguetá, Guaratinguetá, SP, Brazil*
[b] *Fundação Oswaldo Aranha, Volta Redonda, RJ, Brazil*
[c] *Usina Presidente Vargas, Companhia Siderúrgica Nacional, Volta Redonda, RJ, Brazil*
[d] *Universidade Federal de Itajubá, Itajubá, Brazil*

HIGHLIGHTS

- Numerical modeling of natural gas flow through an industrial burner was performed.
- Standard, RNG, Realizable $k-\varepsilon$, and Reynolds Stress Model (RSM) have been used.
- The considered models represent the experimental conditions.

ARTICLE INFO

Article history:
Received 7 November 2013
Accepted 14 February 2014
Available online 4 March 2014

Keywords:
Industrial combustion
Process simulation
Numerical analysis
Industrial burner
Natural gas

ABSTRACT

This paper presents numerical modeling of a turbulent natural gas flow through a non-premixed industrial burner of a slab reheating furnace. The furnace is equipped with diffusion side swirl burners capable of utilizing natural gas or coke oven gas alternatively through the same nozzles. The study is focused on one of the burners of the preheating zone. Computational Fluid Dynamics simulation has been used to predict the burner orifice turbulent flow. Flow rate and pressure at burner upstream were validated by experimental measurements. The outcomes of the numerical modeling are analyzed for the different turbulence models in terms of pressure drop, velocity profiles, and orifice discharge coefficient. The standard, RNG, and Realizable $k-\varepsilon$ models and Reynolds Stress Model (RSM) have been used. The main purpose of the numerical investigation is to determine the turbulence model that more consistently reproduces the experimental results of the flow through an industrial non-premixed burner orifice. The comparisons between simulations indicate that all the models tested satisfactorily and represent the experimental conditions. However, the Realizable $k-\varepsilon$ model seems to be the most appropriate turbulence model, since it provides results that are quite similar to the RSM and RNG $k-\varepsilon$ models, requiring only slightly more computational power than the standard $k-\varepsilon$ model.

1. Introduction

The search for higher energy efficiency of industrial scale combustion furnaces and burners to obtain fuel savings has demanded experimental studies that are complemented with Computational Fluid Dynamics simulations. Industrial burners operate in non-premixed conditions for safety reasons. Fuel and oxidizer enter separately into the combustion furnace and are then mixed and burnt through continuous diffusion only after being discharged from the orifice. Gas fuel is supplied through the orifice and combustion air enters from the surroundings so that the gas can only be burnt within a certain distance from the orifice. Burner designers must determine the correct area of the fuel orifice. For a determined orifice size, there is a specific fuel pressure in order to have the right mix with the combustion air. High fuel pressure can result in soot or flame impingement and low fuel pressure can result in air excess and may not allow the achievement of the required furnace heat load.

Combustion modeling studies have been intensely developed during the last 20 years and the phenomena of flame diffusion is usually treated downstream from the orifice burner. Great

---

* Corresponding author. Av. Ariberto Pereira da Cunha, 333, Departamento de Energia, Guaratinguetá, SP CEP 12516-410, Brazil.
E-mail addresses: joao@feg.unesp.br, joao.a.carvalho.jr@pesquisador.cnpq.br (J.A. Carvalho).

http://dx.doi.org/10.1016/j.applthermaleng.2014.02.036
1359-4311/

Nomenclature

A	geometrical orifice area (m^2)
c	speed of sound ($m\ s^{-1}$)
C_d	discharge coefficient of the orifice (dimensionless)
d	nozzle diameter (m)
D	pipe diameter upstream of the nozzle (m)
f	friction factor (dimensionless)
K	pressure drop coefficient for losses of fittings (dimensionless)
k	turbulent kinetic energy per unit mass ($m^2\ s^{-2}$)
k	ratio of specific heats of the fuel (dimensionless)
L	length of pipe (m)
$\dot{m}$	mass flow rate ($kg\ s^{-1}$)
MW	molecular weight ($kg\ kgmol^{-1}$)
P_1	fuel pressure at upstream from the nozzle ($N\ m^{-2}$)
P_2, P_b	fuel pressure at downstream from the nozzle ($N\ m^{-2}$)
Q	volumetric flow rate ($m^3\ s^{-1}$)
$\overline{R}$	universal gas constant ($J\ kgmol^{-1}\ K^{-1}$)
T	temperature (K)

Greek symbols

α	port angle in a burner nozzle
β	ratio of orifice diameter to pipe diameter
Δ	difference in a quantity
ε	turbulent energy dissipation

Subscripts

1	nozzle entrance
2	nozzle exit

Abbreviations

ICEM CFD	a meshing software
CFD	computational fluid dynamics
DO	discrete ordinate method
EDM	Eddy-Dissipation Model
LFM	Laminar Flamelet Model
PDF	Probability Density Function
RSM	Reynolds Stress Model
WI	Wobbe Index
WSGGM	weighted sum of gray gases model

attention has been given to investigating the interaction of turbulence combustion and its consequences, including deciding which turbulence model is the most appropriate depending on the specificity of the application. Usually the burner orifice modeling does not take part of the analysis. However, for industrial burners it is very important to have information about the pressure upstream from the burner, since changes in fuel chemical composition can be compensated by altering fuel pressure. Burner nozzles are designed for a certain orifice discharge coefficient obtained from experimental measurements. This coefficient is defined as the ratio of actual flow to the maximum theoretical flow and is normally obtained using empirical correlations based on experimental data, derived under controlled laboratories conditions.

The present paper reports a numerical analysis identifying the most appropriate turbulence model to simulate the flow through an industrial burner orifice, establishing comparisons with the best turbulence models for some combustion applications. Flow rate and pressure at burner upstream were validated by experimental measurements. The results indicate that the standard, RNG, and Realizable $k-\varepsilon$ models and Reynolds Stress Model (RSM) satisfactorily represent the experimental conditions. However, the Realizable $k-\varepsilon$ model seems to be the most appropriate turbulence model, since it provides results that are quite similar to the RSM and RNG $k-\varepsilon$ models, requiring only slightly more computational effort than the standard $k-\varepsilon$ model. The industrial burner is a non-premixed swirl burner from the preheating zone of slab reheating furnaces. It is from Usina Presidente Vargas, a major steelworks that belongs to Companhia Siderúrgica Nacional (CSN), located in the state of Rio de Janeiro, Brazil.

2. Literature review

Few attempts have been made to simulate turbulent combustion with burner orifice flow pattern. Indeed, most of the numerical and experimental works have studied orifice meters, and a few have studied hydraulic orifices. The following presents a list of published literature of correlated papers. The quality of the simulations for orifice flow and the choice of turbulent models are discussed.

Experiments on orifice fluid flow measurements were simulated [1] using data provided by others [2,3]. CFD simulations were validated through pressure drop and energy balance measurements using water as fluid. The standard $k-\varepsilon$ turbulence model was used. A comparison of the numerical and experimental results revealed that experimental data closely agreed with CFD predictions.

Experimental discharge coefficients for flow meters, including orifice plate flow meters, were obtained in order to validate numerical results at low Reynolds numbers [4]. The Realizable $k-\varepsilon$ model was used for turbulence closure. The intent of the study was to present characteristic curves to enable users to better understand the relative differences expected at low Reynolds numbers.

CFD was applied to numerically predict the calibration coefficient of orifice meters in order to ease the laborious experimental procedure of calibration [5]. The methodology satisfactorily predicted the discharge coefficients.

The flow through a circular orifice was investigated using CFD and two turbulence modeling techniques [6]. The standard $k-\varepsilon$ model and the Reynolds Stress Model (RSM) were employed. It was found that the results agreed well with experimental data. However, the RSM was more accurate in the downstream orifice region than the $k-\varepsilon$ model.

A numerical study was conducted to evaluate effects of flow through a simple orifice [7]. The main recommendations are that the grid spacing must be 0.1% of pipe diameter upstream of the plate and the use of high-order differencing schemes in order to calculate pressure loss correctly. The standard $k-\varepsilon$ model was used. It agreed sufficiently with experimental data, but the authors recommended the use of other turbulence models or modification of the $k-\varepsilon$ model to improve performance.

Another study [8] proposed to relax the square root relation commonly used by international standards to determine the flow rate through a specific discharge coefficient value. The resulting power law relation was shown to improve accuracy. In addition to the experimental data, evidence was also obtained by performing numerical simulations. The standard $k-\varepsilon$ turbulence model was used.

A single empirical formula to model the flow through hydraulic orifices was proposed [9]. It makes use of a linear relation for small pressure differences and the conventional square root law for turbulent conditions. Simulation results have proved to be accurate.

The effect of contaminated orifice plates on the discharge coefficient was investigated [10]. Experimental work was