Alexandria Engineering Journal (2016) 55, 2309–2320

HOSTED BY

ELSEVIER

Alexandria University

Alexandria Engineering Journal

www.elsevier.com/locate/aej
www.sciencedirect.com

ORIGINAL ARTICLE

Numerical modelling of unsteady flow behaviour in the rectangular jets with oblique opening

James T. Hart [a], **Md. Rezwanul Karim** [a,b], **Arafat A. Bhuiyan** [a,b], **Peter Witt** [c], **Jamal Naser** [a,*]

[a] *Faculty of Science, Engineering and Technology, Swinburne University of Technology, Hawthorn, VIC 3122, Australia*
[b] *Department of Mechanical and Chemical Engineering, Islamic University of Technology (IUT), Gazipur 1704, Dhaka, Bangladesh*
[c] *CSIRO, Division of Minerals, Clayton, VIC 3168, Australia*

Received 21 January 2016; revised 2 May 2016; accepted 11 May 2016
Available online 2 June 2016

KEYWORDS

Recessed slot-burner;
Tangentially fired boiler;
Rectangular jet;
Transient

Abstract Vortex shedding in a bank of three rectangular burner-jets was investigated using a CFD model. The jets were angled to the wall and the whole burner was recessed into a cavity in the wall; the ratio of velocities between the jets varied from 1 to 3. The model was validated against experimentally measured velocity profiles and wall pressure tapings from a physical model of the same burner geometry, and was generally found to reproduce the mean flow field faithfully. The CFD model showed that vortex shedding was induced by a combination of an adverse pressure gradient, resulting from the diffuser-like geometry of the recess, and the entrainment of fluid into the spaces separating the jets. The asymmetry of the burner, a consequence of being angled to the wall, introduced a cross-stream component into the adverse pressure gradient that forced the jets to bend away from their geometric axes, the extent of which depended upon the jet velocity. The vortex shedding was also found to occur in different jets depending on the jet velocity ratio.

1. Introduction

Stable combustion in a tangentially-fired boiler is achieved by orienting the burner jets so as to induce a swirling vortex in the central region of the boiler, providing enhanced mixing and extending the residence time of the fuel to ensure complete burnout. A burner set may contain several burners located in a vertical plane, with a single burner consisting of a primary fuel/air nozzle, sandwiched between secondary air nozzles above and below. The nozzles are rectangular and there is significant separation between them.

In gas fired and black-coal fired boilers the flames are anchored to the nozzle and most of the combustion occurs in the near field of the burner-jets [1–4]. In the near-field of burner jets in a tangentially fired lignite boiler the lignite particles undergo pyrolysis and volatile matter is driven from the particle, but only a limited amount of combustion occurs; therefore, turbulent mixing in the near-field is less crucial. The main aims were to heat the lignite and air by mixing with entrained hot furnace gases and to deliver it to the correct location in the centre of the furnace. Near-field aerodynamics is still important for entrainment and for ensuring that the jets

* Corresponding author.
E-mail address: jnaser@swin.edu.au (J. Naser).
Peer review under responsibility of Faculty of Engineering, Alexandria University.

http://dx.doi.org/10.1016/j.aej.2016.05.008
1110-0168 © 2016 Faculty of Engineering, Alexandria University. Production and hosting by Elsevier B.V.

reach the centre of the furnace at the correct location and with sufficient momentum for generating the required swirl.

Stage II boilers of the Yallourn W power station in the Latrobe Valley, Australia, feature burner nozzles that are recessed in the boiler walls, which permit the boiler size to be reduced while maintaining the correct flame length. Upon commissioning, this burner design was found to operate in a highly unsteady, unpredictable and unsatisfactory manner, impacting significantly on furnace performance. Under certain operating conditions the fuel jets were found to turn so far away from their intended trajectories that they even impinged on the walls adjacent to the burner, causing excessive fouling. This burner design is the object of the current study.

The level of combustion in the near field is low enough that aerodynamics are sufficiently decoupled from the effects of intense chemical reaction and radiative heat transfer, which would otherwise alter the physical properties of the flow. Therefore, isothermal modelling can reasonably be expected to give a good indication how the jets from different burner geometries deliver the fuel stream and mix it with the surrounding gases within the furnace.

Previous investigations by the boiler operator [5,6] in which isothermal, scaled-down physical models were made of several burner geometries, provided measurements of velocity made with a Pitot tube and measurements of the static pressure on the recess walls. Three-dimensional effects are very important in such complex flows, and if modelled correctly, the full flow field prediction from a computational fluid dynamics (CFD) model can provide more insight into the burner aerodynamics than physical modelling alone. The aim of this research was to develop a good understanding of the complex flow patterns that were developed as a result of recessing the burner nozzles in the furnace wall. This paper highlights the unsteady modes of the jets peculiar to this burner configuration and uncovers the cause of the unsteadiness. This knowledge will be valuable in developing future designs and in modifying existing ones.

2. Numerical models and procedures used

The CFD code CFX was used to model the fluid flow, which was subsonic, isothermal, single phase and fully turbulent. The resulting simplified Reynolds Averaged Navier–Stokes (RANS) system of equations solved in this numerical model was for a constant density, constant temperature and transient flow:

$$\partial\rho/\partial t + \nabla.(\mathbf{U}) = 0, \tag{1}$$

$$\partial\rho\mathbf{U}/\partial t + \nabla\cdot(\rho\mathbf{U}\otimes\mathbf{U}) = \nabla\cdot(\sigma - \rho\overline{\mathbf{u}\otimes\mathbf{u}}), \tag{2}$$

where σ is the stress tensor

$$\sigma = -p\delta + \mu(\nabla\mathbf{U} + (\nabla\mathbf{U})^{\mathrm{T}}). \tag{3}$$

Here ρ is the fluid density, $\mathbf{U} = (U, V, W)$ is the mean velocity vector, p is the pressure, μ is the molecular viscosity and $\rho\overline{\mathbf{u}\otimes\mathbf{u}}$ is the Reynolds stress term.

Closure of (2) by calculation of $\rho\overline{\mathbf{u}\otimes\mathbf{u}}$ was achieved using the Shear Stress Transport model of Menter [7], which combined the ability of the $k\varepsilon$ model in modelling free stream turbulence with the ability of $k\omega$ in modelling flow near the wall, in a relatively inexpensive turbulence model. This model has been shown to predict separated flows well. Where the grid resolution was insufficient to implement the $k\omega$ model, wall functions were applied based on the approach of Ref. [8].

A finite-volume coupled solver was used to solve the equations on an unstructured hexahedral mesh. The coupled solver used an Incomplete Lower Upper (ILU) factorisation technique, combined with an algebraic multigrid technique [9] to accelerate convergence, and Rhie–Chow interpolation [10] to solve pressure and velocity on a co-located mesh.

Temporal discretisation was achieved using second-order backward-Euler time differencing with fixed time steps of 1×10^{-5} s. Spatial discretisation of the advection terms used a second order scheme based on [11]. Each time step converged mass and momentum to more than 0.1% based on maximum residuals.

2.1. Model geometry and boundary conditions

The burner was based on the 1/30th scale isothermal physical models of burners in the stage II boilers at Yallourn W from Ref [5], Fig. 1. This 1/30th scale was chosen in order to get detailed information on both mean velocity and turbulent fluctuating components in near and far region of jets as presented by Yan and Perry [12,13]. The burner comprised a nearly square primary jet nozzle flanked above and below by rectangular secondary jet nozzles. The jets first discharged into a small recess in the main wall and then into a large open space. The walls of the recess were divergent at an angle of 10° and the geometric axis of the jets made an angle of 60° with the boiler wall. A small step was placed between the nozzles and the sidewalls of the recess. Geometrically similar jets exhibit similar hydrodynamic behaviour at Reynolds numbers above 1×10^4. Reynolds number is the product of hydraulic diameter of the jet, gas velocity and density over the viscosity of the fluid. Reynolds number in both the furnace and model burners was above 1×10^5; therefore, this model is expected to accurately reproduce the near-field development characteristics of burner-jets in a real furnace [14]. A comparison between the aerodynamic properties of jets in the physical model and the real furnace is shown in Table 1. The dimensions of the cross section of the primary jet and the secondary jet are given in Table 1. The hydraulic diameter (D), which is the diameter of a round nozzle with the equivalent cross-sectional area to the primary nozzle, is 0.0327 m.

For each jet the physical model used a section of ducting, which extended 50 hydraulic diameters (1.65 m) upstream from the nozzle and was fed from a plenum chamber, to achieve a developed flow at the nozzle. The CFD model assumed the jets discharged from a wall into a large open space, with Dirichlet constant pressure boundary conditions used at the open boundaries. These were placed far away from the jet itself, approximately 30 hydraulic diameters in the stream-wise direction, and 15 diameters in the cross-stream directions. Using a symmetry boundary condition on the primary jet centre plane halved the computational burden; the validity of this assumption was confirmed by performing one simulation of the full domain, which showed that no difference existed. Dirichlet boundary conditions were set on the inlet for all jets by specifying a flat velocity profile and turbulence quantities were set based on 1% turbulence intensity, which is appropriate for a flow coming from a plenum chamber. In furnace operation the momentum ratio between the secondary and primary jets