Minerals Engineering 56 (2014) 102–108

Contents lists available at ScienceDirect

Minerals Engineering

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

CFD modelling of fluid flow in a Peirce–Smith converter with more than one injection point

Aaron Almaraz [a], César López [a], Isaac Arellano [a], Miguel A. Barrón [b], David Jaramillo [a], Fidel Reyes [c], Gabriel Plascencia [a,d,*]

[a] CIITEC – IPN, Cerrada Cecati s/n, Col. Sta. Catarina, 02250 México, DF, Mexico
[b] Departamento de Materiales, UAM-Azcapotzalco, Av. San Pablo 180, Col. Reynosa-Tamaulipas, 02200 México, DF, Mexico
[c] Departamento de Ingeniería Química Metalúrgica, Facultad de Química, UNAM Edificio D.C.U., 04510 México, DF, Mexico
[d] Materials Science & Engineering, University of Toronto, 184 College St., M5S 3E4 Toronto, ON, Canada

ARTICLE INFO

Article history:
Received 18 April 2013
Accepted 2 November 2013

Keywords:
CFD
Copper converter
Gas injection
Kutateladze number
Bubbling to jetting transition

ABSTRACT

Copper converting is mainly conducted in Peirce–Smith converters. Extensive work has been conducted to understand fluid flow phenomena as air is injected into molten mattes. High momentum must be transferred from the gas to the melt in order to refine the metal. In this work, we present a CFD analysis of gas injection with one and three tuyeres. Results show that by increasing the number of injection points, the flow pattern within the converter change considerably. Such changes result in the development of large recirculation zones and localised eddy formation. Additionally, it was found that the gas plumes in the melt are asymmetrical thus flow paths constantly interfere between themselves. Bubbling–jetting transition is found to be better represented by the Kutateladze number.

1. Introduction

Copper conversion is the selective oxidation of iron and sulphur with oxygen enriched air (up to 35 vol.% O_2) injected laterally into a molten matte by means of a series of submerged tuyeres aligned on one side of a cylindrical vessel named converter. Nearly all the copper obtained by smelting and converting is processed in Peirce–Smith (PS) converters. In spite of its high productivity, the PS converter has some technical issues:

- Clogging of tuyeres as a result of accretions growing on the tip of them.
- Erosion on the refractory lining around the tuyeres due to localised thermal gradients.
- Inefficient SO_2 capture in the off gases.

This type of furnace has not experienced significant technological improvements over a century of operation, which opens the door for upgrading it (Navarra and Kapusta, 2009). This can be overcome with the use of computational tools and physical modelling of such reactor.

On the other hand, gas injection is vastly used in process metallurgy; a significant amount of analytical and experimental work has been published on this subject. Extensive reviews were made by Brimacombe et al. (1991), Brimacombe (1996), Kellog and Díaz (1992), Mazumdar and Guthrie (1995), and more recently by Lehner and Samuelsson (2009) and Mackey and Campos (2001). In all these reports it has become evident the need to further explore fluid flow phenomena at operating conditions in order to minimise as possible the extent of refractory wear and accretion growth in PS converters, in addition to splashing and slopping of the bath.

In this paper we present some CFD results of the fluid flow within a converter, after injecting gas with one and three tuyeres. Most of the reports existing in literature refer to a single injection point; however, given the nature of the converter, more information on flow patterns and velocity fields is needed when more than a single injection point is used.

2. Model description and geometries used

Fluid flow is described by Navier–Stokes (NS) equations, which are expressed in vector form as follows:

$$\rho\left(\frac{\partial \boldsymbol{u}}{\partial t}+\boldsymbol{u}\cdot\nabla\boldsymbol{u}\right)=-\nabla p+\mu\nabla^2\boldsymbol{u}+\rho\boldsymbol{g} \quad (1)$$

* Corresponding author at: Materials Science & Engineering, University of Toronto, 184 College St., M5S 3E4 Toronto, ON, Canada.
E-mail address: g.plascencia@utoronto.ca (G. Plascencia).

0892-6875/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.mineng.2013.11.001

where ρ is the fluid density, $\boldsymbol{u}$ is the fluid velocity (vector) (m/s), t is time (s), p is pressure (Pa), μ is the fluid viscosity (Pa s) and g is the gravitational acceleration (m/s^2). To maintain the mass balance in the system, the continuity equation must be solved as well:

$$\frac{\partial \rho}{\partial t} = -\nabla \cdot \rho \boldsymbol{u} \tag{2}$$

which under isothermal conditions can be simplified to:

$$\nabla \cdot \boldsymbol{u} = 0. \tag{2a}$$

In current copper converting practice, turbulence is needed to increase copper matte mixing, thus mass, momentum and heat transfer are also increased. In order to model turbulence properly, the Large Eddy Simulation (LES) model was employed. It is well recognised that LES simulates turbulence in a better way than two-equation models such as the κ–ε. LES is applied to turbulent flows and consists of three steps: (i) filtering of NS equations, to remove small spatial scales, allowing for solving large eddies; (ii) use of a model to solve the stress tensor obtained after the filtering stage and (iii) conduct actual numerical simulations (Ghosal and Moin, 1995). From the previous description, it is obvious that finding a proper filter to the NS equations is paramount in order to achieve good numerical results with LES. The filter selection comes from defining a scale length that can divide eddies into either large or small. The filter size used in this work is equivalent to the size of the computational grid.

Liovic and Lakehal (2012) have developed better modelling procedures that involve the effect of surface tension on multiphase flow. The inclusion of the surface tension effect is quite important, for example, Marín and Utigard (2005) showed the importance of accounting for the surface tension on the fire refining of molten copper, as this property modifies fluid flow and hence the mass transfer rate during refining, since Marangoni flow can be induced.

This new calculation approach is based upon increasing the accuracy of the numerical method by modifying the curvature of the gas–liquid interphase as it moves. The soundness of the resulting simulations is improved as the gas–liquid curvature is smoothed. Also important is the discretization of the bi-phasic curvature, which can lead to considerable errors or increased computational time. In addition to the curvature refinement conditions, the use of more stable filters has also allowed for improving the exactitude of the simulations conducted with the LES model. For our calculation procedure, we added a surface tension term to the NS set of equations, similarly to Marín and Utigard. This particular approach differs from that of Liovic and Lakehal's, in the sense that we have accounted for the surface tension effect directly from NS equations and then using a filter similar to the one described (Liovic and Lakehal), rather than modifying the curvature at the gas–liquid interface, With the aid of such filter, we have been able to obtain reasonable numerical results.

In this work, a force term due to surface tension was added to the right side of Eq. (1), this term is:

$$\boldsymbol{F}_\sigma = \sigma \, \Gamma \, \delta_\sigma \, \boldsymbol{n}_\sigma \tag{3}$$

where σ is the surface tension (N/m) at the melt/gas interphase; Γ is the local curvature of the interphase; δ_σ is Dirac's delta function that localises the surface tension force (Yee, 2010) and n_σ is the interphase orientation. Both Γ and $\boldsymbol{n}_\sigma$ are derived from an indicator function $C(\boldsymbol{x},t)$, which is used to trace points occupied by the gas phase from those occupied by the melt:

$$\boldsymbol{n}_\sigma = \frac{\nabla C(\boldsymbol{x},t)}{|\nabla C(\boldsymbol{x},t)|} \tag{4}$$

and

$$\Gamma = \nabla \cdot \boldsymbol{n}_\sigma = \nabla \cdot \left(\frac{\nabla C(\boldsymbol{x},t)}{|\nabla C(\boldsymbol{x},t)|} \right) \tag{5}$$

the indicator function is filtered by a box type filter:

$$C(\boldsymbol{x},t) = \begin{cases} 1 \text{ if } \boldsymbol{x} \text{ is occupied by gas} \\ 0 \text{ if } \boldsymbol{x} \text{ is occupied by liquid} \end{cases} \tag{6}$$

The indicator function is then represented by:

$$\overline{C}(\boldsymbol{x},t) = \int_D G(\boldsymbol{x}-\boldsymbol{x}') C(\boldsymbol{x}',t)\, dx \tag{7}$$

Due to the incompressibility nature of the flow, this indicator function satisfies Eq. (2):

$$\frac{\partial \overline{C}}{\partial t} + \nabla(\boldsymbol{u}\overline{C}) = 0. \tag{8}$$

Additionally, another model is needed to tackle the multiphase flow in the copper converter. The Volume of Fluid (VOF) formulation relies on the assumption that two or more phases are not interpenetrating. For each additional phase (q), its volume fraction (α_q) is introduced as a variable. In each control volume, the volume fraction of all phases sums up to unity. The tracking of the interface between the phases is accomplished by solving the continuity equation for each phase (Fluent, 2003).

$$\frac{\partial \alpha_q}{\partial t} + \overrightarrow{v} \cdot \nabla \alpha_q = \frac{S_{\alpha_q}}{\rho_q} \tag{9}$$

where $S\alpha_q$ is a source term for the q phase. Finally, the Pressure Implicit Splitting Operation (PISO) algorithm was employed in the transient simulations for the pressure–velocity coupling given that it maintains stability in the numerical scheme despite the larger time steps (Fluent, 2003).

Eq. (8) is discretized when VOF interface tracking is employed. For more specific details we refer to Liovic and Lakehal's paper.

To better define the liquid–gas interface, the CFD software used in this research uses an interpolation scheme called "Geometric Reconstruction Scheme" (FLUENT, 2003). The geometric reconstruction scheme represents the interface between fluids using a piecewise-linear approach. This scheme is the most accurate and is applicable for general unstructured meshes. It assumes that the interface between two fluids has a linear slope within each cell, and uses this linear shape for calculation of the advection of fluid through the cell faces.

Another possible source of error in numerical simulations is that of numerical diffusion. Numerical diffusion arises from truncation errors that are a consequence of representing the fluid flow equations in discrete form (Fluent, 2003). Among other factors such as the interpolation scheme, the mesh quality and its alignment with the flow, numerical diffusion is inversely related to the resolution of the mesh; therefore, to reduce the extent of numerical diffusion it is necessary to refine the mesh. In this work, to minimise numerical diffusion, a mesh consisting of 6.5×10^5 elements was used for the simulations of one tuyere, whereas for the simulations with three tuyeres, 1.8×10^6 elements were used. The small mesh size employed in the computer simulations ensures a proper bubble capture and resolution of the gas–liquid interface. The absolute numerical diffusion effect was not estimated.

Earlier simulations (Vaarno et al., 1998) with one injection point used a symmetry that did not consider the effect of adjacent plumes on the flow pattern developed by the simulated tuyere. Consequently, it's been assumed that the resulting plume present certain symmetry features that describe a particular flow pattern. However, results from Rosales et al. (1999) show that there is a strong interaction between adjacent tuyeres that modify the gas flow trajectories obtained by a central tuyere. Such interaction results in flow patterns that are not necessarily symmetrical as it can be assumed based on the actual location of the tuyeres on the