

Analysis of multiphase reacting turbulent jets: case study on carbon injection in siderurgic furnaces

Marina Campolo¹, Michele Andreoli¹, Leonardo Tognotti^{1,3}, Alfredo Soldati^{1,2}

¹Centro Interdipartimentale di Fluidodinamica e Idraulica & ²Dipartimento di
Energetica e Macchine, University of Udine, 33100 Udine, Italy

³Dipartimento di Ingegneria Chimica, chimica industriale e scienza dei materiali,
University of Pisa, 56126 Pisa, Italy

We present an original numerical approach developed to investigate multiphase reacting turbulent flows by discussing an example application in which the injection performance of a new carbon particle injection system for siderurgic applications (More s.r.l.[®]) is to be evaluated. Factors controlling transport, burn-out and devolatilization of carbon particles inside the oxidizing, high temperature environment of the electric arc furnace forbid a precise experimental analysis, and numerical simulations are used to test and characterize injector performances. State of the art numerical techniques are used to characterize the fluid-dynamics and chemico-thermal environment seen by carbon particles, and *ad-hoc* research tools (Lagrangian tracking routines and complex chemistry schemes) are used to reproduce precisely carbon consumption due to thermally and chemically controlled kinetics. Analysis of factors controlling injector performances, identification of most critical configuration of the injector in the furnace and a conservative estimate of the injection yield of carbon particles for different size and quality of carbon powders are obtained at a reasonable cost.

1. Introduction

Multiphase reacting jets are common in the industrial practice and yet often represent critical steps in production since dispersion of the reacting phase may impact significantly on process efficiency. The complex chemico-physical processes controlling transport and reaction of dispersed species make the experimental analysis of these systems extremely difficult: experiments can seldom be performed under controlled conditions, and qualitative comparisons of performances can not be used to improve the physical understanding of phenomena and to drive toward process performance optimization. Computational analysis offers as a cost-effective alternative by which (i) performances can be tested under controlled conditions from the early stages of design onward, and (ii) the effect of many parameters can be considered at a reduced cost with respect to traditional experimental tests.

In this work, we present the numerical methodology developed and adopted to evaluate performances of a "new concept" powder-injection system designed for the steel-making industry. The new powder injector (More s.r.l.[®]) is designed (i) to work as a burner in the early stages of the melting process and (ii) to inject carbon particles to control the chemical (and fluid dynamic) properties of the slag in the later stages. In the traditional design practice, these tasks are performed separately by a burner and a high velocity lance and the injection efficiency of the lance is controlled by the radial spreading of the air jet, which disperses carbon particles into the high temperature furnace environment. In the "new-concept" injector, the oxygen jet issued by the burner

is re-designed to promote the injection of carbon particles. This is obtained by the coaxial arrangement of streams: the inner air jet carrying the carbon particles is surrounded by an annular coaxial high velocity oxygen jet and by an outer coaxial flux of methane. Annular nozzle sections are designed to obtain sonic flow for methane and supersonic flow for oxygen. The supersonic oxygen jet (i) accelerates the particle laden air stream up to very large velocities and (ii) reduces particle radial dispersion, confining and focusing particles during their flight toward the slag. These two effects are expected to increase carbon particle injection yield. Yet, since the probability of burning and devolatilization may increase significantly for particles entrained by the oxygen-rich driving jet, reducing in turn the injection yield, an *a priori* evaluation of injection performances of the new injector is necessary.

The strategy adopted to set up the numerical experiments is based on the precise fluid dynamic characterization of the reacting environment coupled to the use of comprehensive detailed kinetic schemes for carbon particles reaction (Falcitelli et al., 2002). The methodology works as follows: (1) we use state of the art numerical techniques to characterize the flow field generated by the coaxial supersonic jet issuing in the furnace, simulating the transport of the main chemical species and the chemical reactions necessary to get a realistic picture of the chemico-thermal environment, potentially contributing to carbon particle oxidation and devolatilization; (2) we use Lagrangian tracking to derive detailed information about the environment seen by particles traveling inside the furnace; (3) we use these detailed information to build a simplified fluid dynamic model of the reacting environment (Reactor Network Analysis, RNA) in which more complex chemistry models for carbon particles are implemented. The coupling of basic combustion modeling by the flow solver and ideal chemical reactor networks (RNA) has already been reported in previous literature (Niksa and Liu, 2002, among others). The present work represents a further contribution in this field, demonstrating that CFD+RNA modeling methodology is mature for process studies of industrial hot reacting systems.

2. Problem and data

Figure 1 shows the geometry analysed in this work together with a close-up of the injection device. The injector is installed in a 120 tons capacity electric arc furnace (EAF)(diameter 6.9 m and height 3.640 m). The furnace is equipped with three electrodes (102 MW power) and four injectors. Reference working conditions are: methane mass flow rate, 80 Nm³/h at 288.8 K; oxygen mass flow rate, 2900 Nm³/h at 289.02 K; air mass flow rate, 80 Nm³/h at 288 K, and carbon particles mass flow rate, 27 kg/min.

Injection efficiency is evaluated for two different distances of the injector from the slag ($L_1=977$ mm and $L_2=1377$ mm), three different particle size distributions (RRA: $D_{min}=0.106$ mm, $D_{max}=2.$ mm, $D_{ave}=0.295$ mm and spreading exponent $n=2.378075$; RRB: $D_{min}=0.1$ mm, $D_{max}=3.$ mm, $D_{ave}=1.85$ mm and spreading exponent $n=3.356697$ and RRC: $D_{min}=0.106$ mm, $D_{max}=2.$ mm, $D_{ave}=0.48$ mm and spreading exponent $n=1.52305$) and two different types of carbon (Coal 1: high volatile, bituminous coal with fixed carbon, 64.4 %, volatile matter 27.6 %, ash 8 %; Coal 2: anthracite with

volatile fraction $m_v/m=7.5\%$ and fixed carbon 87.7). Shape factor for all carbon particles is 0.7.

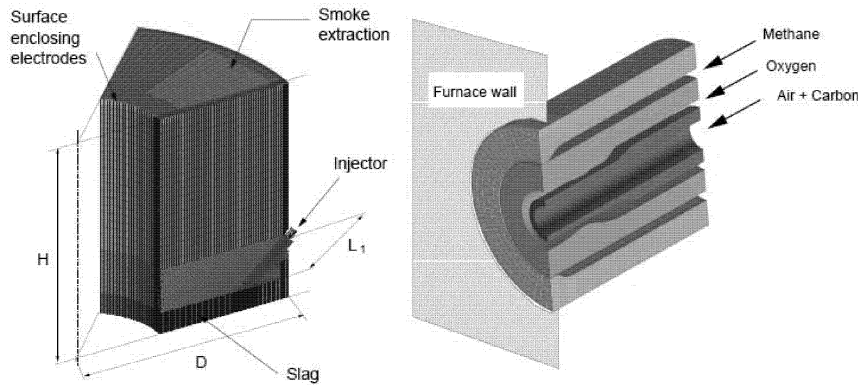


Figure 1: (a) Dimensions and discretization of computational domain. (b) Detail of injection system: particle laden air jet is issued from inner pipe, supersonic oxygen jet is issued from intermediate pipe and sonic flow of methane is issued from outer pipe.

Two external factors influencing carbon particle injection efficiency are considered: (i) air uptake from the top cover of the electric furnace and (ii) radiation effects from furnace walls, electrodes and slag. Due to the complexity of the furnace geometry, we simulate only 45 degrees of the furnace, i.e. the minimal periodical portion extending from the injector symmetry plane to the plane in between two injectors. The liquid steel and electrodes are not included in the computational domain.

The domain is bounded (i) by the slag at the lower side, (ii) by the top wall and by the exhaust extraction section at the upper side and (iii) by a section enclosing the electrodes at the inner radial side.

The computational domain is made of about 340,000 cells. The mesh is finer near the injector to simulate precisely the gradients of velocity, temperature and mass fraction profiles which are largest in this region. Boundary conditions used to set up the numerical simulation are: (i) symmetry plane at the jet middle plane; (ii) no-slip boundary condition for gas velocity at solid walls; (iii) fixed pressure (1 atm) on the side surface; (iv) fixed temperature profile and emissivity values for radiative heat transfer at the walls, at the surface enclosing the electrodes and at the bottom surface; (v) fixed mass flow rate for oxygen, methane and air at the injection point; (vi) fixed outgoing mass flow rate ($65,000 \text{ Nm}^3/\text{h}$) at the smoke exit.

2. Methodology

The problem under study is characterized by (i) multiple streams of different gases (air, methane, oxygen) feeded into an atmospheric air environment; (ii) supersonic/sonic conditions for some of the streams; (iii) chemical reaction between oxygen and methane; (iv) heat transfer/production affecting fluid dynamic behavior of the flow; (v) multiphase flow and complex kinetics for carbon particles injected and moving in a reactive environment (high temperatures and large oxygen mass fraction). We used a

commercial finite volume solver of Navier-Stokes equations (Star-CD[®]) coupled to *ad-hoc* research tools (Lagrangian tracking routines and detailed kinetic models for carbon particles) to describe all these concurrent phenomena precisely and at a reasonable cost:

- Flow field, thermal field and scalar field calculation
Navier-Stokes equations and chemico-thermal enthalpy balance are solved for a carrier fluid (air); k- ϵ turbulence model (Patankar and Spalding, 1972) is used to account for turbulence effects. Conservation of mass is solved for the main chemical species generated by methane oxidation, simplified as a two step process:

$$\text{CH}_4 + 1.5 \text{O}_2 \rightarrow \text{CO} + 2 \text{H}_2\text{O} \qquad \text{CO} + 0.5 \text{O}_2 \rightarrow \text{CO}_2$$
 Reaction rates are calculated using the Eddy Break up model (Magnussen and Hjertager, 1981), with the rate-controlling mechanism being either the chemical kinetics or the turbulent mixing.
- Lagrangian particle tracking
Momentum equation is solved for each single carbon particle considering inertia and drag forces only (Campolo et al., 2005). The drag coefficient is a function of flow regime (particle Reynolds number) and particle shape factor (Crowe et al., 1998). Particle equation is integrated explicitly in time using a time step which is 1/20 of the smallest particle characteristic time, $\tau_p = \rho_p D_p^2 / 18 \mu$. Tri-linear interpolation of fluid velocity values available at grid points is used to evaluate fluid velocity at particle position and to calculate the drag force. The instantaneous fluid velocity at particle position is obtained from the calculated mean flow field and from the local turbulence field (k and ϵ) using the eddy interaction model (Graham, 1998). The size distribution of carbon particles injected into the furnace controls their behavior in the external flow. Therefore, we track many groups of particles with diameter in the range $[D_{\min} \div D_{\max}]$ and for each group, we calculate the radial dispersion and the time of flight distribution. Statistics for the swarm are reconstructed from results obtained for each class of particles weighted by the corresponding mass particle size distribution.
- Reactor network analysis
Statistics derived from Lagrangian tracking are used to characterize the environment seen by carbon particles and to identify an equivalent series of perfectly stirred reactors within which complex kinetics can be handled in a cost effective way (Falcitelli et al., 2002). Volume, residence time, temperature and mass fraction of main species of each reactor depend on the conditions calculated locally by the flow solver. For char combustion, a 0.5 order kinetic rate combined with a diffusion resistance is used. The kinetic rate is a function of burnout. For char devolatilization, empirical correlation for convective heat transfer coefficient, literature data for coal emissivity and conventional modelling of coal devolatilization (Single First Order Reaction, SFOR and Distributed Activation Energy Model, DAEM) are used (see Falcitelli et al., 2002 for details).

3. Results and Discussion

Figure 2 shows (a) velocity isocontours (b) oxygen mass fraction isocontours and (c) temperature isocontours in the jet symmetry plane, for injector position L2. The flow field determines (i) transport and mixing of chemical species issued by the injector and

(ii) acceleration and dispersion of carbon particles. In turn, residence time and chemico-thermal environment seen by carbon particles control their rate of reaction and devolatilization in the EAF.

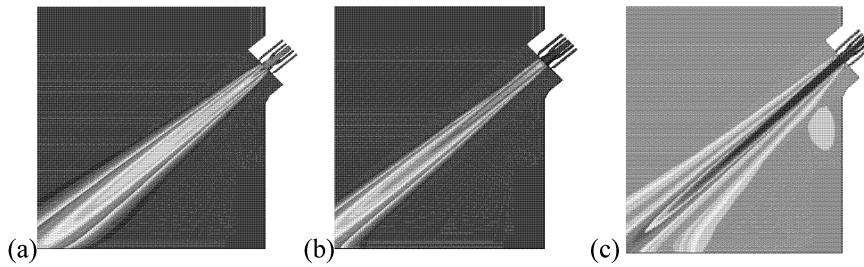


Figure 2: (a) Velocity, (b) oxygen mass fraction and (c) temperature in the jet symmetry plane (injector position L2).

Velocity values range from a maximum of 520 m/s for the supersonic oxygen stream to zero for the outer flow. The high velocity oxygen jet entrains methane (not shown) which mixes rapidly with oxygen and burns and accelerates strongly the inner, slower air jet used to inject carbon particles. High mass fraction of oxygen is found all along the jet trajectory up to the slag. Mass fraction decreases from 1.0 (at the nozzle exit) to 0.65 when the jet reaches the slag. Background temperature in the EAF is 1500°C (1773 K). Larger temperatures are found at the outer surface of the jet. Lowest values are found along the jet axis, associated with the oxygen stream.

Carbon particles injected with the air stream experience a strong acceleration as the air flow is entrained by the supersonic oxygen jet. The acceleration is stronger for smaller particles and yet, due to the short path to travel, particle velocity remains everywhere smaller than the local fluid velocity.

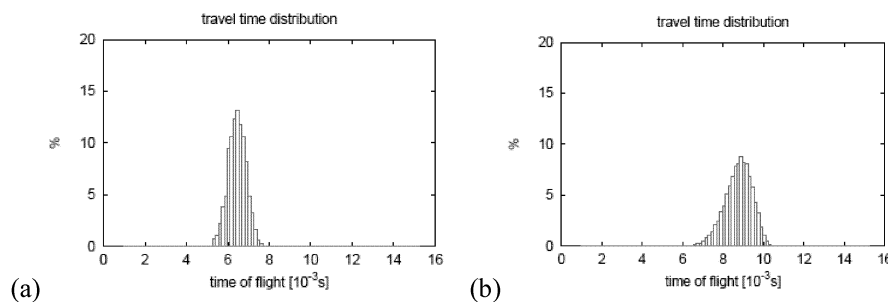


Figure 3: Travel time distribution for particles belonging to (a) RRA distribution and to (b) RRB distribution.

Figure 3 shows the travel time distributions calculated for RR-A and RR-B particles and injector position L2. The average travel time for RR-A particles is smaller than for RRB particles. The travel time indicates the time during which chemical reaction/devolatilization may occur. Results from the Lagrangian tracking are used to calculate also the radial dispersion, which determines the chemico-thermal environment seen by particles in the EAF. Since the jet remains focused, the background fluid sampled by particles moving in the EAF is the oxygen stream issued from the nozzle,

which is colder than the outer fluid in the furnace (see Figure 4 (a) obtained for RRA particles). Bars indicate variations in sampled values due to particles spreading radially into different regions of the jet.

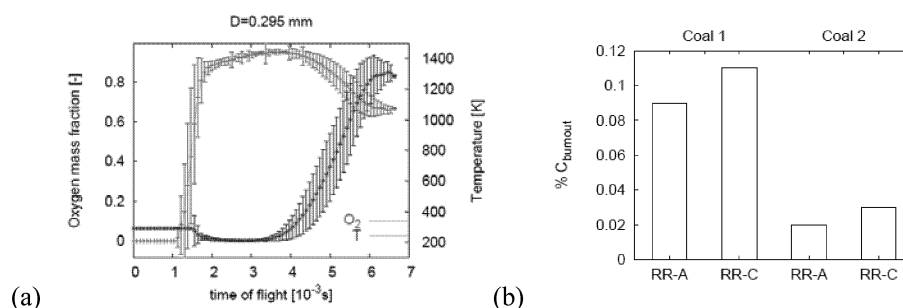


Figure 4: (a) Chemico-thermal environment seen by carbon particles; (b) carbon burn-out for different particle size distributions and different types of carbon.

Particles belonging to RRA distribution move into high temperature regions only during the last 0.001 seconds of their travel. Larger particles (RR-B) stay in high temperature regions of the fluid ($T \geq 500$ K) for the last 0.002 seconds of their travel. The time is anyway too short to produce significant heat transfer to the particles and carbon devolatilization. The oxygen mass fraction seen by the particles shows that they move into an oxygen-rich environment. The mass fraction of oxygen is about 0.9 all along their travel, with negligible differences between distributions RR-A and RR-B.

RNA calculations performed schematizing the particle-laden jet as a series of 80 stirred tank reactors (20 in the low temperature region, 400–500 K, and 60 in the high temperature region, steep increase from 500 to 1400 K) show that devolatilization and oxidation of coal particles starts only in the last millisecond of flight and can be considered negligible, whereas the oxidation process is dominated by thermal effects. Carbon burn-out, i.e. the ratio between the oxidized carbon and the initial carbon content of char particles, calculated by the model is shown in Figure 4 (b) and varies in the range 0.02–0.10 %. Injection efficiency for the injector, given by $\eta = 1 - C_{\text{char}}/C_{\text{char}0} = 1 - \%C_{\text{burnout}}$ is therefore larger than 99%.

4. References

- Campolo M., Salvetti M.V., Soldati A., 2005, Mechanisms for Microparticle Dispersion in a Jet in CrossFlow, *AIChE J.*, 51, 28-43.
- Crowe C., Sommerfeld M., and Tsuji Y., 1998, *Multiphase flows with droplets and particles*, CRC Press,.
- Falcitelli M., Tognotti L., Pasini S., 2002, An algorithm for extracting chemical reactor network models from CFD simulation of industrial combustion systems, *Combust. Sci. Technol.*, 174, 27-42.
- Graham, D.I., 1998, Improved Eddy Interaction Models with Random Length and Time Scales, *Int. J. Multiphase Flow*, 24, 335-345.
- Magnussen, B.F., and Hjertager, B.W., 1981, On the structure of turbulence and a generalised eddy dissipation concept for chemical reaction in turbulent flow, 19th AIAA Aerospace Meeting, St. Louis, USA,.