CHEMICAL ENGINEERING TRANSACTIONS

VOL. 39, 2014

A publication of
AIDIC
The Italian Association
of Chemical Engineering
www.aidic.it/cet

Guest Editors: Petar Sabev Varbanov, Jiří Jaromír Klemeš, Peng Yen Liew, Jun Yow Yong

ISBN 978-88-95608-30-3; **ISSN** 2283-9216

DOI: 10.3303/CET1439290

Optical and Numerical Analysis of Droplet Breakup

Christian Jordan*, Christian Maier, Zsolt Harsfalvi, Bence Kiss, Michael Harasek

Vienna University of Technology, Institute of Chemical Engineering, Getreidemarkt 9/166, A-1060 Vienna
christian.jordan@tuwien.ac.at

The conversion and combustion of liquid alternative reduction agents to injected into an ironmaking blast furnace depends strongly on the achieved droplet size distribution of the spray. Since direct on site investigation of the spray in the blast furnace raceway is not feasible, a lab scale cold model has been constructed. The spray formation has been analyzed using Particle Image Velocimetry (PIV) and High Speed Imaging. Furthermore Computational Fluid Dynamics (CFD) simulations have been carried out and compared to the experimental findings to help understanding the atomization process and to predict the droplet size distributions.

1. Introduction

A simple coaxial injection lance for liquid fuels into a high temperature high speed oxidizer stream has been investigated. Lances of this type are applied e. g. for the injection of liquid hydrocarbons like heavy fuel oil and crude tar from coke production into an iron blast furnace (as practiced e. g. at Blast Furnace A at voestalpine Stahl GmbH in Linz, Austria) or for industrial heating boilers.

The use of such side injection lances in the field of ironmaking blast furnaces has been described in literature to a large extent. Several variants are known, e. g. angled injection lances (Scfasvarf, 1965) or coaxial injection in the tuyere (Sanders, 1963). The use of such lances can be considered as state of the art in liquid injections. Nevertheless previous investigations of liquid jet breakups in high speed flows cover mostly coaxial pressure driven injections (Faeth et. al., 1995), fuel injections into engines (Cao, 2001), high speed wall jets into more or less stagnant gas (Broukal and Hajek, 2011) and wall jet injections into cross flow - different papers consider low cross flow velocities (Ghosh and Hunt, 1998) and moderate and high cross flow velocities (Stenzler et. al., 2006). Correlations for a comparable droplet and flow regime can be found for the coaxial injection (Lewis et. al., 1948) or for a angled side injection (Paloposki and Hakala, 1996) – however, the papers do not consider the influence of the coaxial steam flow of the injection lance. Therefore an in depth-study of the droplet breakup within the turbulent co-flow is carried out. This includes also the effect of variations of the surface tension or the viscosity of the fluid. Investigations have been carried out using a lab scale cold model of a single raceway section of the blast furnace. High speed imaging and subsequent image processing have been applied to evaluate the droplet size distribution at different operating conditions for the injection flow rate and the high speed oxidizer stream.

Computational fluid dynamics (CFD) methods have also been used to model the multiphase liquid – gas flow within the lab test rig. The open source CFD package OpenFOAM (OpenFoam Foundation, 2014) was applied to simulate the jet breakup in the surrounding gas shear flow. Time resolved numerical processing of the spray flow has been done for comparable operating conditions as used for the experimental setup at the Vienna Scientific Cluster 2 (VSC2 - see also: VSC-Team, 2014).

Literature reports various mechanisms (Nagy, 2012) of single droplet breakup occurring in homogeneous shear flow – however, at the lance tip of a fuel oil injection a large velocity gradient causes additional asymmetric shear on the fluid surface. Therefore shear induced entrainment was determined as the major breakup effect.

A simple dimensionless model derived from the measurement results can be used to calculate the typical droplet sizes of blast furnace raceway injections of alternative reduction agents. This ensures the correct computation of the evaporation and combustion locations of the liquid fuels within the blast furnace

Please cite this article as: Jordan C., Maier C., Harsfalvi Z., Kiss B., Harasek M., 2014, Optical and numerical analysis of droplet breakup, Chemical Engineering Transactions, 39, 1735-1740 DOI:10.3303/CET1439290

raceway. A reasonable model of the thermal conditions within the coke bed cavity is the fundamental basis for an accurate prediction of the effects of changes in the operating conditions.

2. Experimental setup

In order to obtain measurement data of the liquid hydrocarbon spray, a lab scale model of one tuyere including the side entry injection lance was constructed (see Figure 1). The scaling factors were calculated based on characteristic dimensionless numbers (especially the Reynolds number, Re, representing the fluid momentum and the Ohnesorge number, Oh, to account for the surface tension effects). The lance tip diameter was selected to be the base length for the size adaption procedure. The apparatus was made from transparent acrylic glass to have full optical access to the spray flow for the visual imaging. PIV measurements (using a Dantec PIV camera system) and high speed camera techniques (using a Photron Fastcam SA-3, similar methods have also been previously applied (Shao et. al., 2003; Jeong et. al., 2007)) have been used to estimate the mean droplet diameter of the liquid spray and initial velocity in the downscaled model. The selected shutter time of about 1/100,000 s required up to 3 kW photo lamps to obtain suitable contrast on the image frames.

To include the effect of the surface tension and the viscosity of the injected fluid, various predefined mixtures of glycerol and water have been applied. Furthermore the operating conditions of the tuyere setup have been varied (blast speed, coaxial lance flow rates, temperature of the injected fluid).

For each trial a suitable number of images has been collected and analyzed using a public domain image processing software “ImageJ” (Rasband, 1997). Macros for automated image handling (area selection, contrast adjustment, droplet identification and sizing) have been prepared (Figure 2). All identified droplet information was accumulated and evaluated.

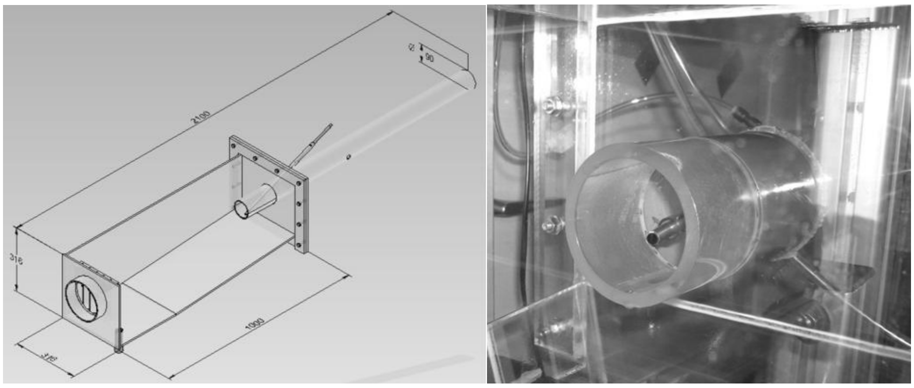

Figure 1: Lab scale test rig for PIV measurements as described earlier (Jordan et. al., 2010), left: overview of the setup, right: detail of the tuyere and injection lance

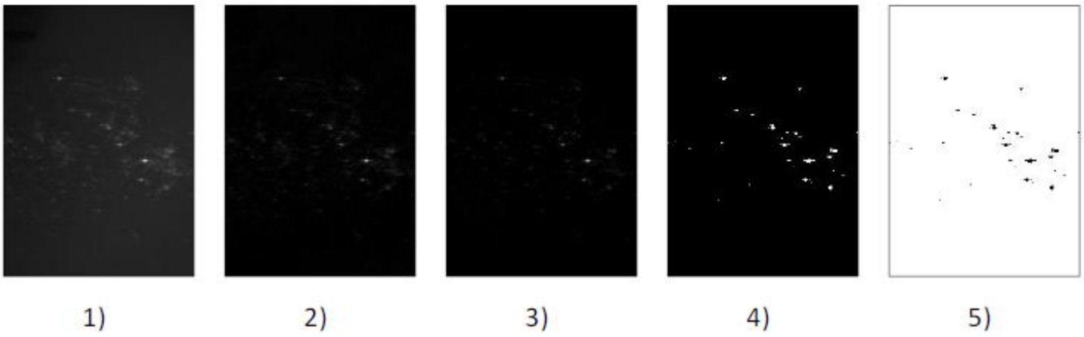

Figure 2: Different steps of the high speed image conversion: 1) original image, 2) background compensation, 3) contour sharpening, 4) conversion from grey scale to black/white, 5) color inversion and droplet sizing

3. Experimental results

Figure 3 gives a first impression of the results from the high speed imaging. A distinct frequency of the droplet shedding vortices could be revealed in the periodically fluctuating liquid jet, varying with the operating conditions.

Best image quality for automated processing was only achieved with low viscosity fluids (high water volume fraction) and high gas speeds. In these conditions full droplet breakup occurred within in the region of 1 – 2 tuyere diameters downstream of the lance tip.

Measurements using high viscosity fluids (modeled with high glycerol concentrations) expectedly show ligament structures, the region of complete droplet breakup was no longer visible inside the investigation window. Furthermore the droplet identification and sizing algorithm was no longer able to deliver any reasonable diameter distribution.

4. Computational fluid dynamics simulations

CFD results with “OpenFOAM” (version 1.7.1 and 2.1) have been calculated in two steps: First the full experimental setup has been modelled with a single phase model only to obtain the overall turbulent flow field of the gas phase. This initial flow field was used as a boundary condition for the more detailed multiphase simulation of the tuyere and the injection lance tip. The full model for the gas phase simulation was discretized with about 2 – 3 million cells, the more detailed model already contained approximately 30 million cells (the characteristic edge length in the lance tip region was found in the range of 100 µm).

Meshing was carried out using “ICEM CFD” or “snappyhexmesh” (part of the “OpenFOAM” distribution). However in a preceding investigation only the mesh produced with “ICEM CFD” showed a suitable and sufficient boundary resolution and grading of the cell sizes.

The single phase calculations used as initial conditions have been carried out with a simple two equation RANS turbulence model (realizable-kε or SST-kω), the multiphase simulations utilized a large-eddy-

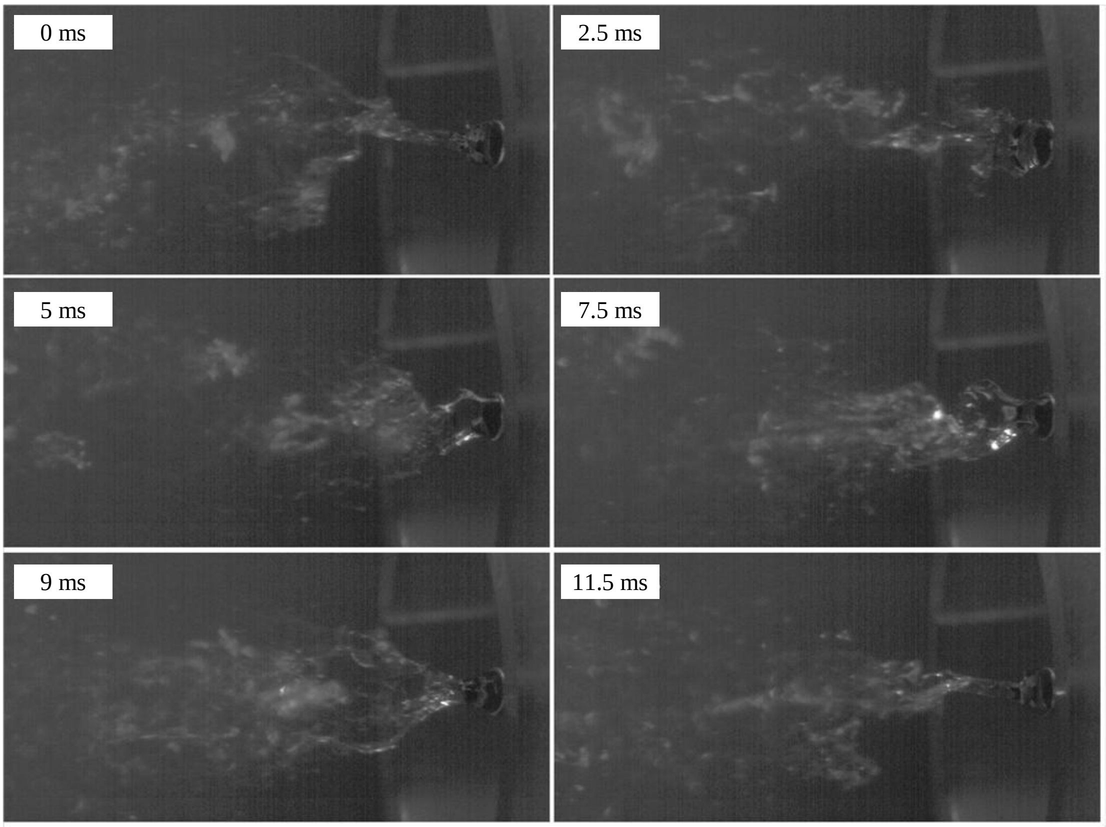

Figure 2: Some example images from the high speed video. The numbers in the individual frames denote the relative run time in ms.

simulation approach in the volume-of-fluid model (LES-VOF) of the interFoam solver. As a standard, 2nd order discretization schemes were used for all flow variables. The flow was considered to be isothermal, for simplicity, incompressible flow

Numerical processing has been done at the Vienna Scientific Cluster 2 (VSC2). Up to 1024 cores (AMD Opteron Magny Cours 6132HE running at 2.2 GHz) have been used for parallel computation, with a time resolution in µs range (adaptive time stepping based on the Courant criterion was used to ensure convergence). The calculation of 1 ms physical time required approximately 2 h wall clock time.

5. Results from CFD

In Figure 3 representative images from a virtual start-up of the injection (activation of the fluid flow) into a constant gas flow are presented. The simulation setup includes water as liquid and 71 m/s gas speed in the tuyere inlet. A 50 % volume fraction iso surface was selected to represent the liquid jet and the droplets. The measured liquid phase distribution from high speed imaging agrees qualitatively with the simulation results.

Similar to the results from the experimental data, the time resolved CFD images reveal a distinct vortex structure which can be used to explain the jet breakup frequency: In Figure 4 periodically moving vortex rings released from the tuyere shell can be seen, decomposing into smaller vortex structures on the right hand side of the image. Figure 5 depicts a contour plot of the vorticity in the symmetry planes, again identifying the vortex ring structure (maxima of vorticity in periodic sequence located in the projected extension of the tuyere shell).

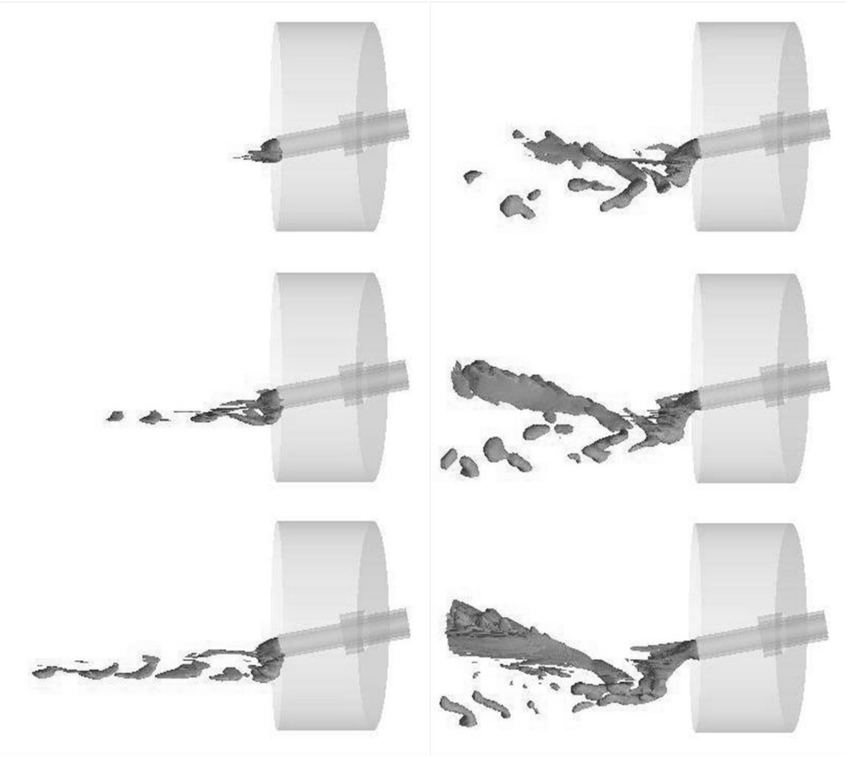

Figure 3: Example frames from the time resolved CFD calculation (base case with a converted blast speed of 71 m/s) – at 1, 2, 4, 6, 8 and 10 ms after activating the injection – showing an iso surface of the liquid volume fraction

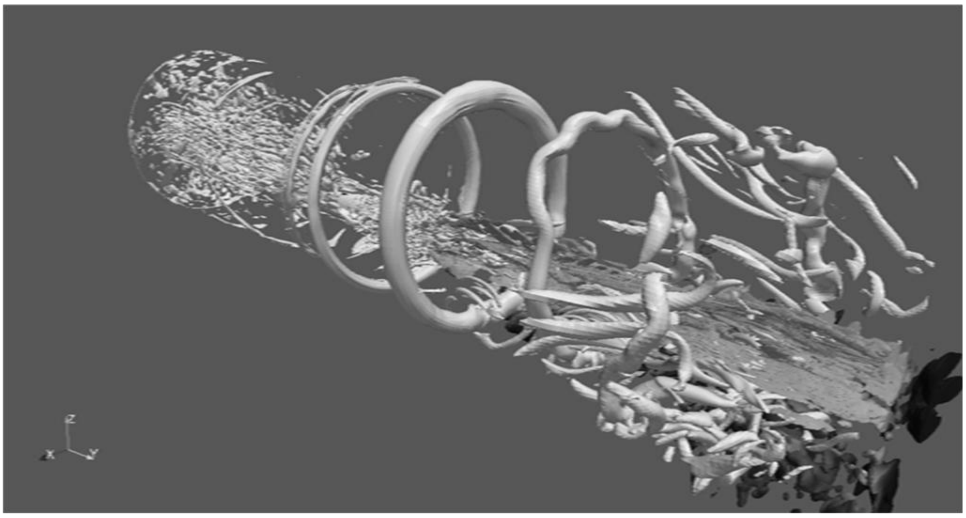

Figure 4: Iso surfaces calculated using the Q-criterion for vorticity (light grey) overlaid with the fluid phase structure (dark grey).

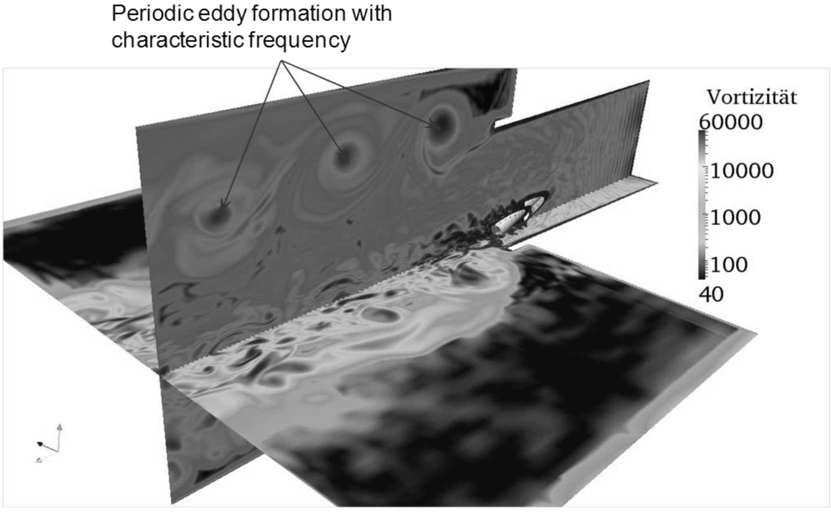

Figure 5: Contour plot of vorticity on the symmetry planes of the simulation geometry.

6. Conclusions and outlook

Reasonable agreement of the measured liquid phase distribution with the simulation results could be reached. The experimental imaging data for low viscosity fluids can be processed to obtain droplet size distributions for the spray flow, which can be used as input parameters for the subsequent full CFD simulation of raceway injections. For high viscosity fluids (and/or low gas velocities) the method does not provide reasonable results.

This also implies that only for low viscosity fluids a simplified discrete particle model (DPM) which usually assumes spherical droplets can be representative for tracking the evaporating and reacting spray droplets. Furthermore only for this flows a rapid jet break-up near to the physical lance tip position can be assumed, otherwise the spray cone angle and the break-up position will need to be shifted into the raceway region. This reduces the heat and mass transfer to and from the liquid, thus a complete reaction within the raceway can not be guaranteed.

A simple dimensionless model derived from the measurement results can be used to calculate the typical droplet size distributions of blast furnace raceway injections of alternative reduction agents. This ensures the correct computation of the evaporation and combustion locations of the liquid fuels within the blast furnace raceway. A reasonable model of the thermal conditions within the coke bed cavity is the fundamental basis for an accurate prediction of the effects of changes in the operating conditions. The results have been analyzed with chemometric methods to propose a correlation for predicting the characteristic droplet sizes within the industrial application.

Acknowledgements

We want to thank our industrial and scientific partners in the FFG/K1-Met framework for their support: Christoph Feilmayr and Christoph Thaler (voestalpine Stahl GmbH, Linz, Austria); Stefan Pirker and Stefan Puttinger (CD-Labor for Particulate Flows, JKU Linz, Austria).

References

Broukal J., Hajek J., 2011, Effervescent Spray Modelling: Investigation of Drop Momentum Models and Validation by Measured Data, Chemical Engineering Transactions, 25, 797-802

Cao J., 2001, On the Theoretical Prediction of Fuel Droplet Size Distribution in Nonreactive Diesel Sprays Journal of Fluids Engineering, 124, 182-185

Faeth G., Hsiang L.-P., Wu, P.-K., 1995, Structure and breakup properties of sprays International Journal of Multiphase Flow , 21, Supplement, 99 - 127

Ghosh S., Hunt J. C. R., Spray jets in a cross-flow Journal of Fluid Mechanics, 1998, 365, 109-136

Jeong H., Lee K., Ikeda Y., 2007, Investigation of the spray characteristics for a secondary fuel injection nozzle using a digital image processing method Measurement Science and Technology, 18, 1591-160

Jordan C., Harasek M., Zauner F., Kirchbacher F., Feilmayr C., Schuster, S., 2010, CFD Analysis of Injection of Heavy Fuel Oil and Plastic Particles into a Blast Furnace Raceway--Estimation of Droplet Size Distribution, Chemical Engineering Transactions, 21, 1087-1093

Lewis H. C., Edwards D. G., Goglia M. J., Rice R. I., Smith L. W., 1948, Atomization of Liquids in High Velocity Gas Streams, Industrial and Engineering Chemistry, 40, 67-74

Nagy J., 2012, Untersuchung von mehrphasigen, kompressiblen Strömungen durch Simulation und Experiment, PhD-Thesis, Institute of Chemical Engineering, Vienna University of Technology (in German)

OpenFOAM-Foundation; Ltd., O. and ESI-OpenCFD, OpenFOAM - The Open Source Computational Fluid Dynamics (CFD) Toolbox, <www.openfoam.org> accessed 10.03.2014

Paloposki T., Hakala J., 1996, Injection of heavy fuel oil into the blast furnace, Proceedings of the Finnish-Swedish Flame Days 1996 (no page numbers given)

Rasband W., ImageJ (U. S. National Institutes of Health, Bethesda, Maryland, USA), <www.imagej.gov/ij> accessed 10.03.2014

Sanders H. R., 1963, US Patent 3,110,584: Liquid fuel injection in blast furnaces.

Scfasvarf J. A., 1965, US Patent 3,209,810: Side-entry fluid fuel injection system for furnaces

Shao J., Yan Y., Greeves G., Smith S., 2003, Quantitative characterization of diesel sprays using digital imaging techniques Measurement Science and Technology, 14, 1110-1116

Stenzler J. N., Lee J. G., Santavicca D. A., Lee W., 2006, Penetration of liquid jets in a cross-flow Begell House, 16, 887-906

Varga C. M., Lasheras J. C., Hopfinger E. J., 2003, Initial breakup of a small-diameter liquid jet by a high-speed gas stream, J. Fluid Mech., 497, 405-434

VSC-Team, 2014, VSC - Vienna Scientific Cluster <www.vsc.at> accessed 10.03.2014