Tribojet, a New and Innovative Device for Atomizing
Liquid Melts

Birgit Matl*, Christian Redl**, Helmut Antrekowitsch*, Alfred Edlinger°

* Christian-Doppler-Laboratory for Secondary Metallurgy of Nonferrous metals,
University of Leoben, Austria

** Böhler Edelstahl GmbH, Research and Development, Kapfenberg, Austria

°Tribovent Processdevelopment Ltd., Bludenz, Austr ia

Summary

The Tribojet is a new innovative atomizing unit for atomizing liquid melts. It is marked by a very low gas consumption and competitive productivity. Computational fluid dynamics (CFD) simulations were done to obtain an in-depth understanding of experimental results and, consequently to allow an investigation of flow characteristics not accesible by measurements. The atomization unit allows to generate a product with a high specific surface and an intensive mixture with the gaseous phase is possible. This leads to the concept of reactive spraying. Furthermore this paper includes a thermodynamic simulation for the slagging of accompanying elements in contaminated cooper melt.

Keywords

Two-fluid atomization, nozzle, computational fluid dynamics

1. Introduction

There are various of different atomizing methods for producing metal powders. Most nozzle designs are specific to each plant and details are proprietary. Decisive for the choice of a particular technique is mainly the scale of operation, but also economical considerations regarding operational costs in order to reach the desired particle size distribution. In literature different ways for the classification of atomizing units can be found. Depending on whether an additional phase is used for disintegration, a single or two-fluid atomization is differentiated. In two-fluid atomization, there are in generally two ways of bringing the liquid and the gaseous phase together, namely external or internal mixing nozzles. Regarding gas demand and velocity, Levebvre [1] defined the air-assist atomizers there a low amount of propellant with supersonic flow disintegrates the liquid melt and the air-blast atomizers, where a large quantity of gas with a relative low velocity (< 100 [m/s]) sprays the second phase.

Maybe the most common way to make fine powders is, on the one hand, the close-coupled technology and, on the other hand, the free-fall technology. In close-coupled or confined technique, the melt is interacting with the gas phase directly at the exit of the nozzle. The short distance between the gas nozzle exit and the melt means that high kinetic energy in the gas can be effectively transferred to the melt. Nevertheless, there is a cooling down due to the expansion that probably determines a freezing at the nozzle's outlet. Negative pressure conditions in the range of the contact between liquid and gaseous phase lead to the deformation of the melting stream and pre-atomization. Furthermore, the suction pressure causes a continuous melting stream.

In case of a free-fall system, the molten metal falls in direction of the gravity field and the gas hits the melting stream in a distance of about 50-200 [mm] under the nozzle`s outlet [2]. This system suffers from a loss of kinetic energy within in the free jet. To achieve the same mean grain size as with a close-coupled unit, the free fall atomizer has to be operated with a higher gas mass flow. In general, the system generates a wider particle size distribution compared with a close-coupled nozzle.

The Tribojet atomization unit is a pilot plant, dimensioned for the capacity of 500 [kg/h] melt used for various investigations in the field of atomization of melts. Characteristic for the Tribojet is that the liquid phase forms a thin continuous film which is disintegrated by a subsonic or supersonic gas stream. Principally, the unit is composed of a mould, a weir tube, an internal (primary) nozzle, and annularly arranged external (secondary) nozzles, which are fed by different propellants. The tube is adjustable in

the axial direction and fixable in the respective axially position. Thus, a gap is created and the melt leaves the atomization unit as a thin continuous film. The internal nozzle disintegrates the film and the molten droplets are caught in the shaft of the plant. The external orifices stabilize the spraying cone and are furthermore used for an additional atomization. Fig. 1 shows the complete system and the atomizer in detail.
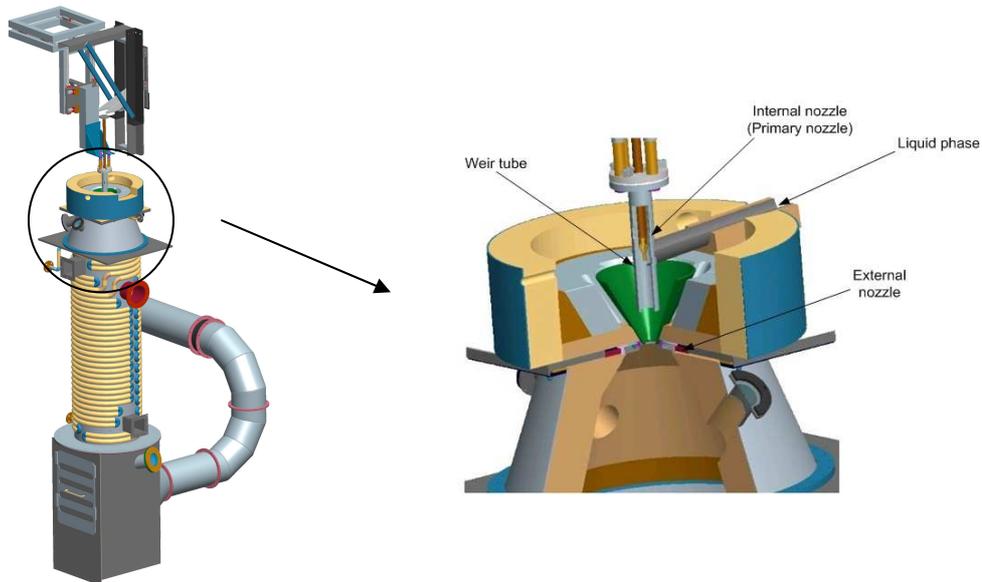

Figure 1: Plant (left) and Atomizer (right) in detail

2. Experimental investigations

The basic investigation regarding the atomization characteristics was done on water models, simplified simulations of the Tribojet pilot plant.

The target of the water model is to analyze the resulting droplet size. Concerning the minimization of the d_{50} (drop mean diameter), the following considerations can be theoretically made:

- The drop diameter decreases with increasing air velocity
- With the increase of the ALR (air-to-liquid ratio), the drop diameter is minimized

- To enable a good atomizing result, the relative velocity between the gas and liquid phase should be high

The second consideration can be successfully verified in tests with water and air on a model which reproduces the Tribojet in the last level of development, including the internal ring (secondary) nozzles. Fig. 2 shows the mean drop diameter depending on different air-to-liquid ratios. For the investigations, a convergent primary (d=7mm) and secondary nozzle (ring nozzle with 24 cylindrical bore holes, d=1.5 mm, angel=10°) were used. An increase of pressure before the primary nozzles, increases the gas flow rate and therefore the ALR [3].

The numerical simulations concerning a two-phase stream support current investigations and give a theoretical prediction of the behaviour of the Tribojet atomizer.
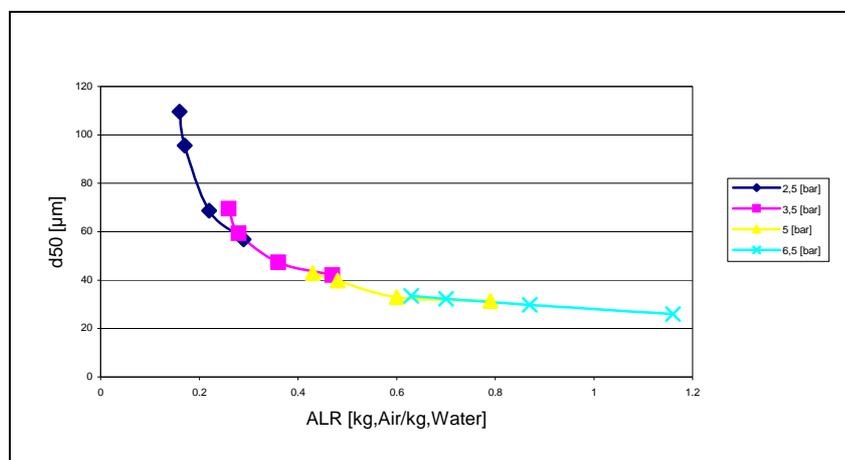

Figure 2: D_{50} depending on the Air-to-liquid ratio [3]

3.   Numerical simulation

The numerical calculation of atomizing processes makes high demands on the simulation software. Multiphase flow problems with gas velocities in the sonic region are serious challenges. As most companies cannot effort to purchase highly specialized software taylored to each problem, general purpose simulation packages are wide spread. As all available computational fluid dynamics (CFD) programs claim

to be applicable to a large range of flow problems it is of great interest when reliable results are delivered for the tasks. Representative for a general purpose CFD code, Fluent 6.1 [4] is used in this study.

Based on the finite volume method [5], the conservation equations for mass, momentum, energy etc. are solved. The gas flow can be mathematically described by the following set of equations:

Equation of mass conservation (Continuity equation)

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \mathbf{v}) = 0 \qquad (1)$$

Equation of motion (Navier-Stokes equation)

$$\frac{\partial}{\partial t}(\rho \mathbf{v}) + \nabla(\rho \mathbf{v}\mathbf{v}) = -\nabla p + \nabla \cdot (\boldsymbol{\tau}) + \rho \mathbf{g} + \mathbf{F} \qquad (2)$$

with the stress tensor given by

$$\boldsymbol{\tau} = \mu \left[\left(\nabla \mathbf{v} + \nabla \mathbf{v}^T \right) - \frac{2}{3} \nabla \mathbf{v} \mathbf{I} \right] \qquad (3)$$

Energy equation

$$\frac{\partial}{\partial t}(\rho E) + \nabla(\mathbf{v}(\rho E + p)) = \nabla(k \nabla T + (\boldsymbol{\tau} \cdot \mathbf{v})) \qquad (4)$$

In the equations above, ρ represent the fluid density, $\mathbf{v}$ velocity, p pressure, $\rho \mathbf{g}$ the graviational bodyforce, μ the molecular viscosity, $\mathbf{I}$ the unit tensor, E internal energy, and T temperature, respectively.

For the numerical studies, the ideal gas law was applied to determine the density of the air used in the equations above. The transport coefficients such as heat capacity c_p and molecular viscosity μ are assumed to be independent from pressure and

temperature. This simplification is acceptable for the first preliminary studies, but can be dropped later on.

There are several approaches in order to track the distribution of the liquid phase. In the Volume-Of-Fluid(VOF)-Method [6], which was used in this study, a transport equation for the volume fraction is solved additionally to eq. 1 - 3. In this specific case, a volume fraction of 1 means that a computational cell is completely filled with air, whereas 0 denotes a cell occupied by the liquid phase. If the value is between 0 and 1, the cell contains the interface between the phases. The interface is represented by a piecewise linear function using the geometric-reconstruction scheme available in Fluent.

The VOF Method is based on the assumption that all considered phases are non-mixable. The motion of all phases is described by one single equation of motion (eq. 2).

Based on the experiences [7] gained from atomizing experiments using water and air in a Tribojet model, a few process-influencing parameters can be identified:

- The gas pressure at the primary nozzle

- The state of the weir tube slits (open or close)

- The filling height of the smelting

- The height of the weir tube opening, through which the melt is entering

- The distance between the primary nozzle and the melt entry

In numerical studies an air-water system corresponding to the experiments is determined.

As a first test the air flow through the primary nozzle was considered. As shown in from fig. 3, the simulation yields higher gas flow rates compared to the experiment [3]. The difference is in an acceptable range considering non ideal experimental conditions.
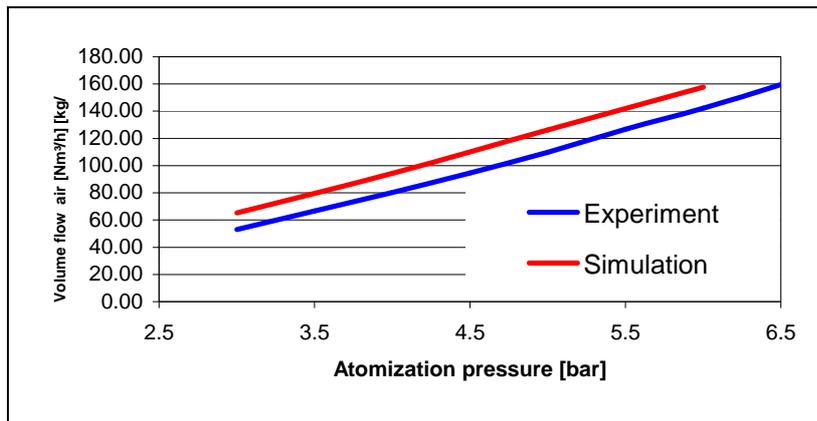
Figure 3: Gas flow rate through the primary nozzle

For the two phase flow simulations, the convergent primary nozzle is taken into account, whereas the secondary ones are being neglected at the moment. The filling height of water in the mould is 150 [mm]. The weir tube is lifted for 1 [mm] to allow the water to enter the atomizer. The pressure before the primary nozzle is 2.2 [bar]. Resulting from the experiments, it seems that a distance of 40 [mm] between the primary nozzle and the melt entry yields a good operating performance. Therefore this distance is also inserted in the calculations.

A two-dimensional axi-symmetric geometry made of about 27.000 computational cells is used. The average length of the cell faces is about 10^{-4} [m]. Due to the strong coupling between flow velocity, density, pressure and temperature in compressible flow, the required computational timestep must be small and is in the range of about 10^{-6} [s].

Experimental observations indicate that the melt is sucked toward the nozzle and squirts upwards into the weir tube if the atomizer is operated with closed slits. This can be easily explained by the fact that a substantial negative pressure within the weir tube develops, if the tube is closed. This phenomenon was captured very well in the simulation. Fig. 4 exemplarily shows an instantaneous snap shot of the phase distribution in case of closed weir tube slits. It can be seen very clearly that there is no liquid film formation at the outlet nozzle. The liquid water is colored in blue, whereas red represents the gaseous phase.
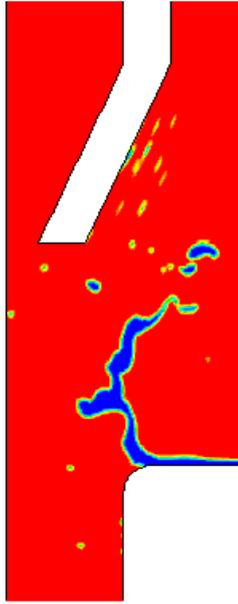
Figure 4: Phase distribution in case of closed weir tube slits

On the other hand, if the slits are kept open during the operation, a pressure compenstation between the weir tube and the outer atmosphere takes place. Therefore, the squirting of the melt into the tube is reduced and the formation of a liquid film is enhanced. An instantaneous phase distribution and the time averaged phase distribution are depicted in fig. 5. Unfortunately this formation proceeds not ideally, but the film is also lifted off from the outlet nozzle (marked by the arrow in fig. 5 on the right side). Nevertheless, a disintegration of the liquid can be observed.
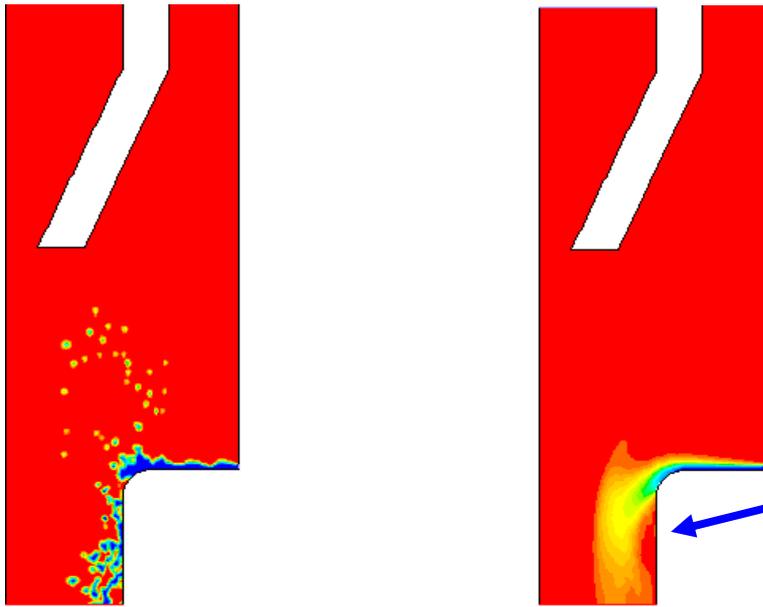
Figure 5: Phase distribution in case of open weir tube slits (left: instantaneous, right: time averaged)

The authors believe that the apparent droplet size in the simulation is a quantity which should by examined with care, as it heavily depends on the size of the computational cells. In fact, the smallest scales which can be resolved are obviously bigger than the cell dimensions. Nevertheless, it is very important to note the tracking capabilities of the VOF model using the geometric reconstruction scheme.

4. Application areas

The main target which leads to the development of the atomization unit is to produce granulated slag for the cement industry. The aim is the spraying of melted blast furnace slag to amorphous micro particles, whereas the slag enthalpy is used for preheating the atomization gas.

The principle for slag atomization leads to preliminary tests for spraying liquid metals, thin and molten copper. Water vapor, air and optional inert gases are used for

spraying. Table 1 shows data from preliminary investigations where 120 [kg] copper have been atomized with air as spraying gas.

Mean diameter d_{50}	[mm]	0,065
Specific surface S_v	[m²/cm³]	0,103
Sauter mean diameter d_{32}	[mm]	0,0581

Table 1: Preliminary test, sprayed copper

5.   Prospects

The presented atomization unit allows to spray liquid melts and to generate a product with a high specific surface. Furthermore an intensive mixture between gaseous and liquid phase is possible. This leads to the concept of "Reactive Atomization", whereas a melt is sprayed by an oxidising gas.

From a theoretical point of view, a thermodynamic calculation based on a chemical composition of contaminated copper (table 2), so-called black copper has been done.

%	Cu	Ni	Pb	Sn	Zn	Fe	As	Sb	Ag	Cd
	74.87	5.42	3.81	5.88	2.13	7.51	0.08	0.21	0.076	0.0005

Table 2: Chemical composition black copper

The aim of the simulation was to particularly calculate the selective slagging of the accompanying elements Fe, Sn, Pb, Ni and Zn. Starting from 100 g of metal (Stream 1, T=1100°C) the equilibria were determined in the ran ge of 0 to 30 g in steps of 0.1 g oxygen (Stream 2, T=25°C). The estimation based on adiabatic conditions (ΔH=0).

The calculation was carried out by using the FactSage database FtmiscCu-LQA [8], which is optimized especially for copper, including chemical interactions in the melt regarding the contained elements. The activity coefficient for tin was estimated with FactSage from the binary system Sn/Cu with γ_{Sn}=0.07. In literature, γ_{Sn} in a copper melt is calculated with 0.11, independent from the composition at 1200 °C [9]. Fig.6 shows the dependency between melt, copper in the melt, resulting slag phase, and cuprous oxid in the slag phase.

Due to the slagging of the elements with a high affinity for oxygen like iron and zinc, the total mass of the melt decreases. After an increase of the oxygen content in the melting phase (Fig. 7), tin and nickel start to oxidize at the same time. The activity of nickel in a copper melt is raised and therefore the tendency to a slagging is higher than described through the Richardson Ellingham diagram for metal and metal oxides. Remarkable is that nickel and antimony built a solid compound, which changes its state at an addition of 4 [g] of oxygen. Therefore the Ni content arises (Figure 7). As expected, the slagging of lead starts at the end.
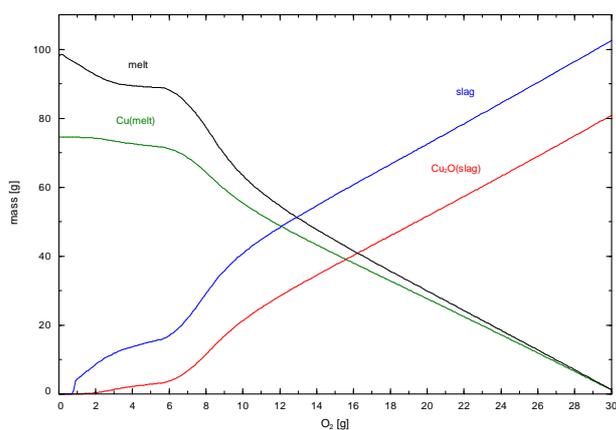

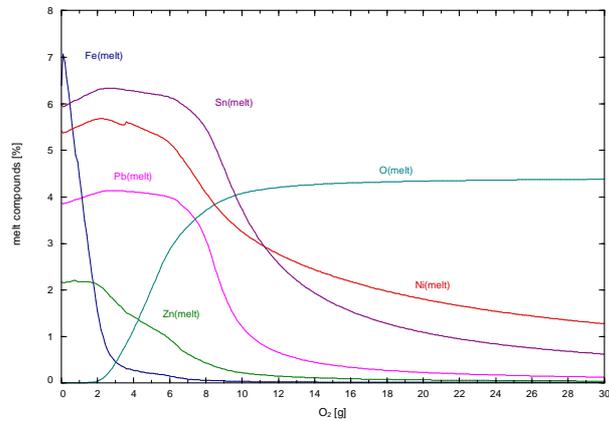Figure 6: Dependency between metal phase, copper in the metal phase, slag, and Cu_2O [8],[10]

Figure 7: Slagging of the accompanying elements, oxygen curve [8],[10]

6. Conclusion

It has been shown that it is possible to simulate trans-sonic multiphase flow problems using a general purpose CFD package. The obtained results are in agreement with experimental observations. In particular, it was possible to confirm the case with open or close wear slits by the numerical simulation. These slits can be also used to transport hot exhaust gases from the heating system to the melt inlet to prevent freezing at the outlet nozzle. In a next step, this concept will be studied numerically. Relating to the thermodynamic simulation, it is sure that the calculation only represents a theoretical point of view, without considerations of the reaction kinetics during the spraying process. In order to have a large surface between the gas and liquid phase, the next steps in optimization the present plant are regarding the minimization of the drop size.

Supplementary preliminary experiments on the water model are planned to determine test parameters for further investigations on the pilot plant. Computational fluid dynamics simulations may be helpful to gain a deeper insight into the interaction between primary and secondary nozzles.

References

[1]   Lefebvre, A. H., Atomization and Sprays, Hemisphere Publishing Corporation, 1989.

[2]   Heck, U. Zur Zerstäubung in Freifalldüsen, Doctor thesis,VDI-Verlag Reihe 7, Nr. 348, Düsseldorf (Germany), 1998.

[3]   Steindl, F., Modellversuche bei der Mikrogranuation von Hochofenschlacke; MSc Thesis, University of Leoben (Austria), 2004.

[4]   Fluent Inc., Fluent Users Guide *6.1*, Lebanon (USA), 2004.

[5]   Versteeg, H. K., Malalasekera, W., An Introduction to Computational Fluid Dynamics – The Finite Volume Metho*d*, Longman, 1999.

[6]   Hirt, C. W., Nichols, B. D., Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, J. Comp. Phys. 39 (1981), pp 201 – 225.

[7]   Kerschbaum, T., Experimentelle Untersuchungen einer Zweistoffdüse zur Schlackenzerstäubung, MSc Thesis, University of Leoben (Austria), 2003.

[8]   Factsage 5.0, Database FtmiscCu-LQA

[9]   Pawlek, F., Metallhüttenkunde,Walter de Gruyter Verlag, Berlin, New York, 1983.

[10]   Factsage 5.0, Database Ftoxid-SLAGA