CFD SIMULATION OF MULTIPHASE MELT FLOWS IN STEELMAKING CONVERTERS

Hans-Jürgen Odenthal, Udo Falkenreck, Jochen Schlüter
SMS Demag AG,
Eduard-Schloemann-Straße 4, 40237 Düsseldorf
Germany
E-mail: hans-juergen.odenthal@sms-demag.com
Web page: http://www.sms-demag.com

Key words: bottom tuyere, combined blowing BOF converter, depth of penetration, mixing efficiency, mixing time, multiphase flow, supersonic top lance nozzle

Abstract. The three-dimensional, transient and non-isothermal flow of melt, slag and oxygen has been simulated for a 335 t combined blowing converter with six-hole top lance and 14 bottom tuyeres. The calculation is based on the Reynolds averaged Navier-Stokes (RANS) equations and the Standard $k$-$\varepsilon$ model. The time-dependent formation of the melt cavities is modeled by the Volume of Fluid (VoF) approach. In order to simulate the stirring gas plumes, individual argon bubbles are released from each bottom tuyere. The bubbles are treated as dispersed phase and modeled using the Discrete Phase Model (DPM). The fundamental flow phenomena such as the penetration of the supersonic oxygen jets, the motion of the phase interfaces, the behavior of the gas plumes and the interaction with the bulk flow as well as the heat transport in the refractory lining are predicted reasonably well. However, the blowing process can only be calculated for a limited process period due to computing-time reasons. Thus, the mixing time, which gives information on the homogenization process, is calculated for pure bottom-blowing. The CFD model provides an efficient tool to describe and further improve the combined blowing process.

1 INTRODUCTION

Today, the metallurgical refining process of hot metal in steelmaking converters is state of the art. The converter is a large vessel with a capacity up to 400 t of melt at high temperatures of 1650 to 1700°C. The converter system mainly consists of a vessel steel shell, refractory lining, vessel protective slag shields, trunnion ring, vessel suspension system supporting the vessel within the trunnion ring, support bearings and the retractable top lance. Three methods to run a converter are possible:

In the first case of top-blowing converters (LD - Linz Donawitz), oxygen at high pressure ($p < 14$ bar) and velocity ($Ma > 1$) is injected downward into the melt using a water-cooled lance with a convergent-divergent (CD) nozzle to produce a supersonic oxygen jet. In most applications, supersonic jets are preferred to subsonic jets due to a higher oxygen entrainment into the melt. The reduction of carbon, phosphorous and other impurity levels is high because
the generated area for the exothermic reaction is large.

In the second case of bottom-blowing converters (OBM - Oxygen Bottom Maxhütte), oxygen and inert gas are injected upward under the melt bath through tuyeres located in the bottom refractory lining. Additives such as pulverized lime and carbon or hydrocarbon fuels such as natural gas or fuel oil can also be injected.

The third case is a combination of top and bottom blowing, which is called combined blowing technology. It is characterized by a multi-hole top lance inducing several spreading supersonic oxygen jets. Simultaneously, inert gas (Ar, N₂) is blown through porous plugs or tuyeres (cooled/uncooled) in the bottom lining to achieve stirring. The oxygen jets and the heterogeneous gas plumes out of the bottom elements force the melt to be agitated and mixed. The mixing efficiency, characterized by the homogenization time after the introduction of additives into the converter, is high. The chemical reaction between melt and slag is fast. The combined blowing technology is widespread and the specific designation differs from one company to another, i.e. LBE - Lance Bubble Equilibrium, LD-KGC - LD Kawasaki Gas Control, K-BOP - Klöckner Basic Oxygen Process, TBM - Thyssen Blowing Metallurgy. However, the process still offers opportunities to further improve the steel quality and plant productivity. For example, the converter geometry, top lance configuration, number, dimension and arrangement of bottom elements as well as the blowing rates affect the flow pattern. These parameters are of decisive importance for the oxidation process.

Important aspects of top-blowing are the transport of oxygen into the reaction zone, the desired formation of a gas-slag-melt emulsion (foaming) and the mixing of the metal bath. The key demands of top-blowing are a dynamic supply to follow the process requirements and a high momentum to maximize jet penetration and bath agitation. The latter demand improves the rate of slag formation, refining reactions and mixing intensity.

Important aspects of bottom-blowing are the metal bath geometry as well as the number, position and gas flow rate of the bottom elements. A certain number of bottom elements can be recommended for any converter, dependent on the maximum flow rate and minimized wear demands. The main objective of the arrangement of the bottom elements is to decrease the C, Mn and P content and lower the Fe content in the slag. The metallurgical improvements of combined blowing are to bring the slag-melt reactions closer to equilibrium and to homogenize both, the temperature and the composition of the melt bath.

The current publications can be roughly divided into those dealing with pure top-blowing processes¹⁻¹⁰ and pure bottom-blowing processes¹¹⁻²⁴, not only for BOF (Basic Oxygen Furnace) and AOD (Argon Oxygen Decarburization) applications, but also for research purposes. Only few reports²⁵⁻²⁸ deal with the simulation of combined blowing technologies. From the literature review, the following aspects can be taken for granted:

- The depth and diameter of the cavity are increased with increasing top-blowing rate and converter back-temperature as well as decreasing bath density and lance height³,⁴,⁷⁻⁹,¹⁰,²⁷. The parameters affect the bath oscillation and direction of splashing⁵,¹⁴,¹⁷.
- The high speed oxygen jets are responsible for the desired foaming process and the undesired slopping and skulking process⁹,²⁹.
- Shrouded nozzles induce a higher depth of penetration than conventional nozzles¹,⁶,⁷,³⁰.
• A nozzle inclination angle greater than approx. 11° prevents the jets from interactions\textsuperscript{5}. Top lances with twisted nozzle tips produce higher slag splashing rates\textsuperscript{10}.

• Shallow melt levels and low amounts of slag decrease slag splashing\textsuperscript{10}.

• For pure top-blowing, the mixing time increases with melt height and lance distance\textsuperscript{3,27}. The larger the surface of the depression, the higher is the rate of oxidation.

• For pure bottom-blowing, the mixing time decreases with increasing melt height, gas flow rate and number of bottom elements\textsuperscript{3,27}. Elements that release swirling gas bubbles provide a better mixing efficiency\textsuperscript{22-24}. Additional top-blowing prolongs the mixing time\textsuperscript{11,17,28}. The distance between bottom elements and converter wall must be large enough to restrict refractory wear.

There are still inconsistencies regarding the arrangement of the bottom elements and the top lance nozzles to each other. While Ajmani et al.\textsuperscript{11} found that an increased off-centre position of the elements increase the mixing time for pure bottom-blowing, Lachmund et al.\textsuperscript{26} report on improved process conditions for such a modification. Anyway, the latter result is found for combined blowing conditions. Another example concerns with the number of bottom elements. Several studies have shown that at a given flow rate, an increased number of bottom elements increases the mixing time. Other studies argue the converse\textsuperscript{17}.

Operational trials relating to the fluid-dynamic behavior of the converter melt are not possible due to the ambient conditions. Nowadays, either physical or numerical simulation is used to analyze splashing and mixing phenomena. Computational Fluid Dynamics (CFD) is able to describe the complex fluid flow in blast and electric arc furnaces, converters, continuous casting facilities and hot rolling mills. The distribution of the flow quantities, i. e. $u_i$, $p$, $T$, $\rho$ and $v$, or species transport can be calculated throughout the domain. Before CFD can be applied, it has to meet the necessary and the sufficient conditions. Necessary means, that the fundamental physics of the flow is reproduced within a needed accuracy. This condition is widely fulfilled, today. Sufficient means, that the time to work through the numerical chain to develop a new product must not exceed the time limit. This condition depends on the complexity of the CFD model, the available computing power and finally on the user.

In the opinion of the authors, a comprehensive CFD model to predict the melt flow in converters does not exist. It is the objective of the present work to develop a CFD model which is able to describe as many flow related phenomena in combined blowing converters as possible and thus helps to clarify the above-described inconsistencies.

2 CONVERTER GEOMETRY

The simulated converter is used for the production of low carbon steel grades. Fig. 1 gives an overview of the computational domain. The converter consists of a 90 mm thick steel shell and refractory linings of different materials and thicknesses. The bath level is $H_m = 1.65$ m corresponding to a capacity of 335 t. The height of the slag (mean components are 50 % CaO, 20 % FeO, 2 % MgO, 16 % SiO$_2$) is $H_{sl} = 0.26$ m. The distance between top lance and melt is chosen to be $H_{tl} = 1.0$ m to better emphasize the flow related effects such as jet penetration and splashing. In order to mix the melt, argon is injected through 14 tuyeres in the converter bottom. The tuyeres are located on two concentric circles. The total mass flow rate of all bot-
tom tuyeres is \( m_{\text{ar}} = 560 \text{ Nm}^3/\text{h} \).

Tab. 1 shows the main dimensions and boundary conditions of the six-hole top lance. Tab. 2 contains the main dimensions of the converter and the properties of the fluid and solid phases.

a) Design of the 335 t converter with steel shell (ST) and refractory lining (RL-1, RL-2, RL-3)

b) Six-hole top lance

c) Location of the inner (R₁) and outer (R₂) bottom tuyeres

Fig. 1: Computational domain of the converter with six-hole top lance and bottom tuyere arrangement

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow rate ( V_{\text{ox}} )</td>
<td>1100 Nm³/min</td>
</tr>
<tr>
<td>Total oxygen pressure ( p_0 ) bar</td>
<td>13.0</td>
</tr>
<tr>
<td>Total oxygen temperature ( T_0 ) °C</td>
<td>20</td>
</tr>
<tr>
<td>Total oxygen density ( \rho_0 ) kg/m³</td>
<td>17.1</td>
</tr>
<tr>
<td>Converter back-pressure ( p_c ) bar</td>
<td>1.2</td>
</tr>
<tr>
<td>Converter temperature ( T_c ) °C</td>
<td>1650</td>
</tr>
<tr>
<td>Number of top lance nozzles ( n )</td>
<td>6</td>
</tr>
<tr>
<td>Nozzle to nozzle angle ( \alpha_n ) °</td>
<td>7.4</td>
</tr>
<tr>
<td>Nozzle inlet diameter ( d_{n,1} ) mm</td>
<td>48.1</td>
</tr>
<tr>
<td>Nozzle throat diameter ( d_{n,2} ) mm</td>
<td>41.5</td>
</tr>
<tr>
<td>Nozzle exit diameter ( d_{n,3} ) mm</td>
<td>58.9</td>
</tr>
<tr>
<td>Nozzle inlet length ( l_{n,1} ) mm</td>
<td>16.0</td>
</tr>
<tr>
<td>Nozzle throat length ( l_{n,2} ) mm</td>
<td>13.5</td>
</tr>
<tr>
<td>Nozzle diffuser angle ( \beta_n ) °</td>
<td>7.4</td>
</tr>
<tr>
<td>Nozzle diffuser length ( l_{n,3} ) mm</td>
<td>135.0</td>
</tr>
</tbody>
</table>

Tab. 1: Main dimensions and boundary conditions of the six-hole top lance, see Fig. 1.b
Tab. 2: Dimensions of the converter and properties of the fluid and solid phases

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Fluid</th>
<th>Solid ST</th>
<th>Solid RL-1</th>
<th>Solid RL-2</th>
<th>Solid RL-3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>melt</td>
<td>slag</td>
<td>oxygen</td>
<td>melt</td>
<td>slag</td>
</tr>
<tr>
<td>height H m</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>nose diameter D₀ m</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>inner diameter D₁ m</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>outer diameter D₂ m</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>lance height Hₙ m</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>charge weight m t</td>
<td>335.7</td>
<td>39.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>charge volume V m³</td>
<td>48.4</td>
<td>11.2</td>
<td>280.4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>melt height Hₘ m</td>
<td>1.65</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>slag height Hₙ m</td>
<td></td>
<td>0.26</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>density ρ kg/m³</td>
<td>6932</td>
<td>3490</td>
<td>variable</td>
<td>5.1×10⁻³</td>
<td>100×10⁻⁴</td>
</tr>
<tr>
<td>surface tension σ N/m</td>
<td>1.54</td>
<td>1.54</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>dynamic viscosity µ kg/ms</td>
<td>5.1×10⁻³</td>
<td>100×10⁻³</td>
<td>0.019×10⁻⁴</td>
<td>1.2×10⁵</td>
<td></td>
</tr>
<tr>
<td>pressure p Pa</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>temperature T °C</td>
<td>1650</td>
<td>1650</td>
<td>15</td>
<td></td>
<td></td>
</tr>
<tr>
<td>thermal conductivity λ W/mK</td>
<td>40</td>
<td>1.7</td>
<td>2.5×10⁻²</td>
<td>43</td>
<td>4.2</td>
</tr>
</tbody>
</table>

2 NUMERICAL MODELS

Two models have been developed to analyze the fluid flow in the converter. The first model is related to the combined blowing conditions and considers the three-dimensional, transient, non-isothermal and three-phase flow of melt, slag and oxygen. The CFD model considers the high speed flow of oxygen out of the top lance and the rising argon bubbles from the bottom tuyeres. If desired, the exothermic reaction between oxygen and melt can be integrated by a User Defined Function (UDF). However, the CFD model is only able to simulate the process for a limited process period of a few seconds. This is caused by the large velocity gradients with velocity components from a few mm/s inside the slag/melt phase up to 550 m/s inside the gas phase. Assuming, that a grid cell has a length of 20 mm, the time step size to resolve the flow field must be about 3.6×10⁻⁵ s. Thus, the computing time for a blowing period of 15 min. would be too long, even in the case of a multi-processing computation.

As a result, the second CFD model considers the above-mentioned effects for pure bottom-blowing. Since the velocity gradients are low, the time step size can be increased. The non-isothermal behavior of the fluid and solid phases is calculated as well.

The CFD simulation is based on the Reynolds averaged Navier-Stokes (RANS) equations and the k-ε-turbulence model with standard values for the constants. Additional transport equations are solved for the calculation of the energy and species distribution. The near wall flow is described by the standard wall function. Because all equations are described in detail in31,32 they are not presented, here.

The time-dependent motion of the slag and melt levels (ox-sl, sl-me, ox-me) is calculated using the Volume of Fluid (VoF) model33. It is a surface tracking method and is applied to
immiscible multiphase flows, where the location of the interface between the phases is of interest. In the VoF approach, an additional transport equation for the volume fraction $\alpha_s$ of the secondary phase is solved. The transport equation for the secondary phase is

$$\frac{\partial \alpha_s}{\partial t} + u_i \frac{\partial \alpha_s}{\partial x_i} = 0.$$  \hspace{1cm} (1)

The volume fraction of the primary phase is $\alpha_p = 1 - \alpha_s$. For the secondary phase applies

- $\alpha_s = 0$: cell is completely filled with primary phase, $\alpha_p = 1 - \alpha_s$,
- $\alpha_s = 1$: cell is completely filled with secondary phase,
- $0 \leq \alpha_s \leq 1$: cell shares both phases.

In a multiphase system with $n$ phases, $(n-1)$ equations are required. The coupling between the momentum conservation and the phases depends on the density $\rho$ and the viscosity $\nu$:

$$\rho = \rho_s \alpha_s + (1 - \alpha_s) \rho_p$$

$$\nu = \nu_s \alpha_s + (1 - \alpha_s) \nu_p.$$  \hspace{1cm} (2)

The free surface is seen as a discontinuity of the material data. Since the problem involves a high momentum transfer from the supersonic gas phase into the liquid phase, the fluid domain is strained with areas of high and low Re numbers. Especially after the start of blow, this aspect leads to convergence problems, so that a user input is necessary.

The exothermic reaction between oxygen and melt is simulated by patching an energy source on the interface. By this, the temperature in each grid cell containing the oxygen-melt interface is increased up to approx. $T = 2600^\circ C$. However, this procedure represents a user input on the solution algorithm and has to be carried out with care.

### 2.1 Top-blowing

Supersonic gas jets impinging onto a slag-melt interface cause a momentum based depression to be formed on the liquid surface. The gas moves radially outward from the impact point along the free surface dragging the slag/melt into motion and induces a circulation of the bulk melt. The mechanisms of surface deformation are known as dimpling, splashing and penetrating\(^{1,6,8,38}\). The formation of these modes depends on the melt, slag and gas composition, the flow rate and the lance height. The jet is mainly responsible for phenomena such as oxidation, foaming, skulлин, slopping, converter oscillation and noise.

A supersonic velocity can only be produced by a CD nozzle. The jet downstream the nozzle is subdivided into potential core, supersonic and subsonic region, Fig. 2. Within the core region ($x_1/d_1 \approx 6$), the velocity is constant. The end of the supersonic region ($x_2/d_1 = 10$ to 20) is defined by $Ma = 1$. Downstream, the velocity is subsonic. The jet interacts with the environment and produces a region of turbulent mixing. The entrainment process increases the mass flow rate and the jet diameter and decreases the mean axial velocity as the distance from the nozzle exit increases. The impact force on the slag-melt surface is reduced with increasing lance height. The jet length and the spreading angle are affected by the gas condition $p_c$, $T_c$ and $p_e$ in the converter. For example, as the ambient temperature $T_e$ increases, $x_2/d_2$ increases as well. In most steelmaking applications, jets with $2.0 < Ma < 2.4$ are used.
In order to validate numerical results, it is prerequisite to analyze the compressible flow inside the CD nozzle using the isentropic theory for ideal gases. The theory assumes that the gas entropy throughout the nozzle is constant. For the design point, the back-pressure is equal to the exit pressure ($p_c = p_1$). In this case, the nozzle is most efficient. If $p_c < p_1$, the flow rate could further be increased by increasing $p_c$. If $p_c > p_1$, the flow will be choked. The pressure $p$, temperature $T$ and density $\rho$ at any location inside the nozzle can be calculated by

$$\frac{p}{p_0} = \left(\frac{1}{1 + \frac{k-1}{2}Ma^2}\right)^{k-1} \quad (3), \quad \frac{T}{T_0} = \frac{1}{1 + \frac{k-1}{2}Ma^2} \quad (4), \quad \frac{\rho}{\rho_0} = \left(\frac{1}{1 + \frac{k-1}{2}Ma^2}\right)^{\frac{1}{2}}. \quad (5)$$

For $Ma = 1$ and $\kappa = 1.4$, the critical values are $p^*/p_0 = 0.528$, $T^*/T_0 = 0.833$ and $\rho^*/\rho_0 = 0.634$. To compute the design point, the mass flow rate $\dot{m}$, the total pressure $p_0$, the total temperature $T_0$ and the back-pressure $p_c$ must be known. The throat area is

$$A^* = \dot{m} \left[2p_0\rho_0 \left(\frac{p^*}{p_0}\right)^{\frac{k}{k-1}} - \left(\frac{p^*}{p_0}\right)^{\frac{k+1}{k}}\right]^{0.5}. \quad (6)$$

The exit surface $A_1$ and exit Mach number $Ma_1$ are determined from equations (7) and (8):

$$A_1 = A^* \left[\left(\frac{2}{k+1}\right)^{\frac{k+1}{2}} \left(\frac{p_0}{p_0}\right)^{\frac{k+1}{k}}\right]^{0.5}. \quad (7)$$

$$Ma_1 = \left[\frac{2}{k+1} \left(\frac{p_0}{pc}\right)^{\frac{k-1}{2}} - 1\right]^{0.5}. \quad (8)$$

Although the isentropic theory is used to design CD nozzles, it does not predict choke reflections and compression/expansion waves. It must be noted that each nozzle can only be designed to fit a certain value of $\dot{m}$, $p_0$, $T_0$ and $p_c$. However, the blowing rate in a converter changes significantly. The simulation procedure of top lance blowing is as follows:

- The two-dimensional (axisymmetric), compressible, steady-state flow of oxygen in- and outside the nozzle is simulated to yield the basic flow effects. The results are validated by the isentropic theory.
This information is used to calculate the three-dimensional, compressible, transient oxygen flow of a nozzle which is aligned perpendicular to the slag-melt interface. Anyway, using the ideal gas law in conjunction with the VoF model leads to a sensitive system of transport equations.

To manage the problem with the compressible phase, the latter is seen as incompressible ($\rho_{ox} = \text{const.}$). The exit conditions $p_1$, $T_1$ and $\rho_{ox}$ are modified so that the normalized momentum force $\rho_{ox}u^2$ along the jet axis is similar to the compressible solution. Thus, the depth of penetration and bath agitation are comparable to the real top-blowing process. The depth of depression is compared with results from the literature. The error due to the increased oxygen density is negligible since $\rho_{ox}/\rho_{at} < 1$ and $\rho_{ox}/\rho_{m} < 1$.

### 2.2 Bottom-blowing

Bubble columns are contactors in which the disperse gas phase moves relative to the continuous liquid phase inducing an overall flow pattern. The advantages of bubble columns are the absence of moving parts, leading to easier maintenance, high interfacial areas and transport rates between the phases and good heat transfer characteristics. The complex behavior of bubble columns reactors affects the operation and performance and has been widely investigated. In metallurgical reactors, large individual bubbles are released from the tip of the tuyere at low gas flow rates (bubbling regime). The bubbles are rapidly decelerated and expanded by a factor of approx. five. In between the release of two bubbles, tuyere flooding can occur. If the flow rate increases, the gas will form an oscillating jet that may cause back-attack (transition regime). At higher flow rates, the near orifice jet becomes more stable (jetting regime). At some distance above the orifice, the jet breaks-up into a swarm of rising bubbles and forms a cone-shaped gas plume. Since the bubbles disperse and coagulate, the size distribution within the plume differs from individual bubbles. In general, it is not possible to numerically resolve individual gas bubbles above all tuyeres in space and time. Therefore, the gas plumes in the converter are simulated using the Discrete Phase Model (DPM). In addition to the transport equations for the continuous phases, the bubbles are treated as dispersed phase and calculated in a Lagrangian reference frame. The DPM approach predicts the trajectory of a bubble by integrating the acting forces of inertia, drag and buoyancy. Turbulent fluctuations are superimposed by a stochastic tracking approach known as Discrete Random Walk (DRW) model. The turbulent dispersion is predicted by integrating the trajectory equations for a bubble, using the instantaneous flow field, along the particle path during the integration. By computing the trajectory for a sufficient number of tries, the random effects of turbulence on the bubble dispersion are accounted for. In DPM modeling, the dispersed phase must not exceed 10% by volume of the mixture in any domain. The present model considers transient coupling phenomena. Bubble break-up, coalescence and growth mechanism are not considered in the present investigation. To model the gas plume, simplifications had to be made. The rising velocity $u_b$ and the shape of a bubble in melt depend on the bubble diameter $d_b$:

$$d_b = \left[ \frac{\sigma_{buv}}{\rho_m g} \right]^2 + 0.03325 \left( V_{at}^2 d_{buv} \right)^{0.867} \right]^{0.167}$$  \hspace{1cm} (9)
Here, \( d_{tuy} \) is the diameter of the tuyere, \( \sigma \) is the surface tension and \( \dot{V}_{tuy} \) is the tuyere gas flow rate. The rising velocity \( u_b \) is

\[
u_b = \left( \frac{2\sigma}{\rho_m d_b} + 0.5gd_b \right)^{0.5}.
\]

(10)

All quantities in equations (9) and (10) are given in SI units. With \( \dot{V}_{tuy} = 40 \text{ Nm}^3/\text{h} \) and estimated values for \( p_{tuy} \) and \( T_{tuy} \) at the outlet of the tuyere, the mass flow rate is \( m_{tuy} = 0.02 \text{ kg/s} \). In this case, \( d_b = 65.2 \text{ mm} \) and \( u_b = 0.57 \text{ m/s} \). The lower the mass flow rate, the smaller are \( d_b \) and \( u_b \). Rising velocities of bubbles in melt are in the range of 0.5 to 1.5 m/s. In turbulent melt flows, the stability of a bubble can be diminished due to collision effects.

In this investigation, gas injection is realized by a star-shaped bubble distribution with \( n_b = 61 \) individual bubbles, Fig. 3. The bubble diameter \( d_b \) has a linear size distribution, starting with \( d_{b,max} \) in the centre to \( d_{b,min} \) at the outer diameter. To sum up the infinite flow rates \( \dot{m}_{i,m} \) yields the total flow rate \( \dot{m}_{tuy} \) of the tuyere. Each bubble is released from a fixed location \( x_i \) and with a certain velocity \( u_i \), diameter \( d_b \), temperature \( T_b \) and flow rate \( \dot{m}_{i,m} \). The process of argon injection is time-dependent. The bubbles are trapped at the oxygen-slag and the oxygen-melt interface using a UDF.

| max. bubble diameter \( d_{b,max} \) | m | 0.052 |
| min. bubble diameter \( d_{b,min} \) | m | 0.002 |
| diameter increment \( \Delta d_b \) | m | 0.010 |
| injection surface diameter \( d \) | m | 0.150 |
| injection surface increment \( \Delta \xi \) | m | 0.015 |
| number of bubbles per row \( i \) | - | 11 |
| tuyere diameter \( d_{tuy} \) | m | 0.08 |
| tuyere mass flow rate \( \dot{m}_{tuy} \) | kg/s | 0.02 |

Tab. 3: Flow conditions for the bottom tuyere

The drag laws for spherical particles can not be used in the present case. Apart from the overall bubble deformation, playing an important role for the bubble movement, the surface tension is important, because it causes an internal circulation which affects the bubble behavior. As the Weber number increases, an initially spherical bubble is distorted. Only for very low values of \( d_b \), the bubble can be seen as rigid sphere (Stokes law). Bubbles with \( d_b > 2 \text{ mm} \) almost appear irregularly, getting an ellipsoidal shape and finally become cape shaped. The irregular moving surface affects the direction of motion. These bubbles have a constant drag coefficient \( c_D \). In this work, \( c_D \) is modeled according to Bröder\textsuperscript{12}:
The objective of the tuyere arrangement is to minimize the time to achieve complete mixing. To determine this period, a scalar transport equation for the mass concentration, given as

\[
\frac{\partial (\rho c)}{\partial t} + \nabla \cdot (\rho \mathbf{u} c) = \frac{\partial}{\partial x_i} \left( D_{\text{eff}} \frac{\partial c}{\partial x_i} \right),
\]

is solved throughout the computational domain, but only for the melt phase. This is, because the melt can enter other fluid domains as well. $D_{\text{eff}}$ is the effective mass diffusion coefficient.

The turbulent mass transfer is modeled using the concept of Reynolds’ analogy to turbulent momentum transfer. The mass diffusion is dominated by the turbulent transport and described by the turbulent Schmidt number $Sc_t = 1$. According to the Stokes-Einstein relation, the laminar mass diffusion coefficient of liquid steel is $D_l \approx 4.4 \cdot 10^{-9} \text{ m}^2/\text{s}$. The turbulent heat transfer is treated similarly with the turbulent Prandtl number $Pr_t = 0.8$.

The mixing process is evaluated by introducing a small amount of tracer into the domain at steady-state flow conditions $(t = t_0)$ and monitoring its dispersion, either at sampling positions or by volume-averaging. Since the injection point influences the mixing process, different locations have been tested. If the tracer is released from two or more domains inside the bubble columns, the computing time will be low because the tracer is transported by forced convection. Releasing the tracer from other starting points will delay the convective transport. The mixing time $\Theta_{95}$ is defined as the time taken for the uniformity $U$ to reach 0.95

\[
U = 1 - \frac{\bar{c}_0 - c_i}{\bar{c}_0}. \tag{14}
\]

In equation (14), $\bar{c}_0$ is the equilibrium concentration and $c_i$ is the concentration at a certain location and some instant in time. The concentration is determined at various locations in the converter and averaged to obtain $\Theta_{95}$. According to Bothe et al.\textsuperscript{39} the mixing efficiency $M$ is

\[
M = 1 - \frac{\sigma}{\sigma_0}. \tag{15}
\]

The standard deviation $\sigma^2$ and the deviation $\sigma_0^2$ at $t = t_0$ are

\[
\sigma^2 = \frac{1}{V} \int_V (\bar{c}_0 - c_i)^2 \, dV, \quad \sigma_0^2 = \bar{c}_0 (c_{\text{max}} - \bar{c}_0) \text{ with } \bar{c}_0 = \frac{V_{\text{sc}}}{V_{\text{sc}} + V_m}. \tag{16}
\]

In eq. (16), $V_m$ is the total melt volume and $V_{\text{sc}}$ the volume filled with an initial blob of tracer ($c = c_{\text{max}}$). In this study, the mean concentration is $\bar{c}_0 = 0.27\%$. 

\[
a) \quad \text{Re} \leq 1.5 \quad \text{(spherical)} \quad c_D = 16 / \text{Re} \\
b) \quad 1.5 \leq \text{Re} < 80 \quad \text{(spherical)} \quad c_D = 14.9 \text{Re}^{-0.78} \\
c) \quad 80 \leq \text{Re} < 700 \quad \text{(ellipsoid)} \quad c_D = 48 / \text{Re}(1 - 2.21 \text{Re}^{-0.5}) \\
d) \quad 700 \leq \text{Re} < 1530 \quad \text{(ellipsoid)} \quad c_D = 1.86 \cdot 10^{-15} \text{ Re}^{4.756} \\
e) \quad 1530 \leq \text{Re} \quad \text{(cape style)} \quad c_D = 2.61
\]
2.3 Numerical procedure

The two-dimensional, compressible, steady-state nozzle flow is calculated using the coupled solver with implicit linearization. A second-order upwind scheme is used to discretize the governing equations. The grid is axisymmetric, consists of 0.3 Mio. cells and is adapted to pressure gradients to capture compression/expansion waves. To get faster convergence, the segregated solver is used initially and changed to coupled after the flow field has converged. Pressure work, kinetic energy and viscous dissipation effects are included when solving the energy equation. The double precision solver is applied to minimize rounding errors.

The three-dimensional, incompressible, transient, multiphase flow in the converter is computed using the segregated solver with implicit linearization. One half of the converter is approximated by a block-structured grid with hexahedral cells. The number of cells varies from 0.2 to 1.0 Mio. dependent on the case of application. The grid density is high in regions of large gradients, i. e. below the lance tip and at the free surface, Fig. 4. The blocks are spread from the lance tip to the bath level. All boundary conditions are chosen to meet the real converter process. Pressure boundary conditions are used at the nozzle exit (pressure inlet) and at the converter mouth (pressure outlet). Compared to velocity boundary conditions, this choice reduces the overestimation of turbulence in areas of large pressure and velocity gradients. For the temporal discretization of the mass, momentum and energy equation, an implicit, and for the VoF model, an explicit time-marching method is used. The solution of the diffusive terms is performed with the central difference scheme. The convective terms, except that one in the VoF equation, are solved with upwind schemes of first-order accuracy. Due to numerical instabilities it was necessary to use the geometric-reconstruction-scheme according to Youngs for the convective term of the VoF model. This high-order discretization assumes that the interface between two fluids has a linear slope within each cell and uses the shape for calculation of the advection of fluid through the cell faces. The pressure-velocity correction is done with the PISO procedure, the PRESTO scheme is used for pressure discretization. The calculations have been carried out using FLUENT 6.3 and executed on a Linux cluster with a maximum of 5 parallel processes.

Fig. 4: Block-structured hexahedral grid and cells in the exit region of the six-hole top lance
3. RESULTS

3.1 CD nozzle

Fig. 5 shows the simulated distributions of Ma and T for the design point of the CD nozzle. The exit plane characteristics are $Ma_1 = 2.13$, $p_1 = 1.26$ bar, $T_1 = -117.6^\circ$C, $\rho_1 = 3.14$ kg/m$^3$. The calculated length $x_2/d_1$ of the supersonic region as a function of the converter back-pressure $p_c$ can be expressed as

$$x_2/d_1 = -12.41 \ln p_c + 172.31 \quad \text{for } 0.8 \text{ bar} \leq p_c \leq 4 \text{ bar}. \quad (17)$$

Equation (17) is valid for an undisturbed jet, entering an atmosphere of $T_c = 1650^\circ$C. For the given configuration, it is $x_2/d_1 = 27.2$. This value is higher than for a jet entering an atmosphere at room temperature ($x_2/d_1 = 10$ to 20). The density $\rho_c$ of the ambient gas (oxygen) is temperature-dependent and affects the expansion of the jet. Increasing the ambient temperature $T_c$ increases the length of the supersonic region $x_2/d_1$ because the ambient density $\rho_c$ is reduced. Böttcher$^1$ indicates that $x_2/d_1$ increases by a factor of three in a hot environment ($x_2/d_1 = 12|T_c = 27^\circ$C, $x_2/d_1 = 35|T_c = 1727^\circ$C). The ratio $\rho_{ox}/\rho_c$ of the jet and the ambient density should be high to enable a large depth of penetration.

Fig. 5: CFD simulation of the CD nozzle; Mach number and temperature distribution for the design point

Fig. 6 and Tab. 4 compare the results of the CFD simulation with the isentropic theory, equations (3) to (8). The numerical simulation fits well the analytic solution. The relative error of the critical values (index *) is smaller than that one of the exit values (index 1). The distribution of $Ma$ and $T$ outside the nozzle indicates the presence of compression and expansion waves. The geometry of the nozzle has not been fully adapted to the ambient conditions.

The oscillation of $p$, $T$, $\rho$ and $Ma$ outside the nozzle is characteristic for either under-expanding ($p_1 > p_c$) or over-expanding jets ($p_1 < p_c$). Fig. 7 shows the influence of the back-pressure $p_c$ on the Mach number distribution.

For $p_1 < p_c$ (Fig. 7a) oblique shocks are released from the nozzle corner and the effective surface of the jet decreases. Downstream, the jet pressure increases up to values higher than the back-pressure. The shocks are reflected as expanding waves at the free jet boundaries and the jet pressure decreases. A complex form of overlapping compression and expansion waves occurs (diamond pattern). The periodic compression and expansion is repeated until the spreading jet shear layer reaches the supersonic region and the jet becomes subsonic.

For $p_1 > p_c$ (Fig. 7c, d) the jet is expanded behind the orifice. A fan of expanding waves is
released from the nozzle corner, reflected as overlapping compression waves until the jet becomes subsonic. A further decrease of $p_c$ induces a Mach disk as a result of increasing pressure difference between emerging jet and ambience. The Mach disk is a discontinuous change of the flow conditions from the hypersonic to the subsonic case.

Fig. 6: CFD simulation of the CD nozzle; Mach number and temperature distribution for the design point

<table>
<thead>
<tr>
<th></th>
<th>CFD</th>
<th>isentropic theory</th>
</tr>
</thead>
<tbody>
<tr>
<td>throat *</td>
<td>exit 1</td>
<td>throat *</td>
</tr>
<tr>
<td>$p/p_0$</td>
<td>0.5288</td>
<td>0.0972</td>
</tr>
<tr>
<td>$T/T_0$</td>
<td>0.8369</td>
<td>0.5306</td>
</tr>
<tr>
<td>$\rho/\rho_0$</td>
<td>0.6315</td>
<td>0.1839</td>
</tr>
<tr>
<td>$Ma$</td>
<td>0.995</td>
<td>2.127</td>
</tr>
<tr>
<td>$m$ in kg/s</td>
<td>4.27</td>
<td></td>
</tr>
<tr>
<td>$</td>
<td>\Delta Ma/Ma</td>
<td>$ in %</td>
</tr>
<tr>
<td>$</td>
<td>\Delta m/m</td>
<td>$ in %</td>
</tr>
</tbody>
</table>

Tab. 4: Comparison between CFD simulation and isentropic theory; CD nozzle for the design point, $p_0 = 13$ bar, $T_0 = 20^\circ$C, $\rho_0 = 17.07$ kg/m$^3$ according to Tab. 1

Ma

a) $p_c = 1.5$ bar (over-expansion, $p_1 < p_c$)

b) $p_c = 1.2$ bar (near design point)

c) $p_c = 1.0$ bar (under-expansion, $p_1 > p_c$)

d) $p_c = 0.8$ bar (under-expansion, $p_1 > p_c$)

Fig. 7: CFD simulation of the CD nozzle; influence of the converter back-pressure $p_c$ on the jet
CFD satisfactorily resolves the supersonic flow phenomena. However, care must be taken to design a nozzle because over- and under-expansion as a reason of varying converter blowing conditions occur. Another point is the generation of noise, which can be regarded as the scattering of the incident oblique shocks by the transient motion of the turbulent mixing layer.

3.2 Top-blowing

The top-blowing process is transient and difficult to analyze. Steady-state flow conditions do never occur, but after a certain period of time, as the shape and depth of depression and the circulation of the liquids have been fully developed, quasi-steady-state flow conditions develop. The process needs a few seconds to become stable. Fig. 8 illustrates the impact of a high speed gas jet \((u_i = 530 \text{ m/s}, Ma = 2.21)\) which is aligned perpendicular to the slag and melt phase. The lance height is \(H_l = 1.0 \text{ m}\), the slag height is \(H = 0.26 \text{ m}\). The freely-selectable CFD parameters of the incompressible approach have been modified until the depth of depression is comparable to the predicted depth by Koria and Lange. The authors carried out experiments on molten steel to determine the penetrability of impinging single and multi-hole gas jets. The depth \(H_{cav}\) and diameter \(D_{cav}\) of the cavity are a function of the gas supply pressure, the impinging distance and the number and inclination angle of the nozzle. A covering slag layer is not considered during their tests. Assuming that \(H_0 = 1.0 \text{ m}\), \(d_{n,3} = 58.9 \text{ mm}\), \(p_0 = 13 \text{ bar}\), \(p_c = 1.2 \text{ bar}\), \(\alpha_n = 0^\circ\), the derivation by Koria and Lange leads to \(H_{cav} = 0.71 \text{ m}\) and \(D_{cav} = 2.1 \text{ m}\). Since the depth of depression is well represented by the simulation, the diameter of the depression is smaller \((D_{cav} = 0.8 \text{ m})\). However, the flow related effects are predicted reasonably. The spatial extension of the turbulent jet is similar to a supersonic jet, but without depicting supersonic effects such as shock waves. The oxygen-slag and slag-melt interface are deformed, slag is radially displaced by the jet and surface waves are formed. Gas is entrained into the melt phase. The depth of penetration reaches a maximum at \(t \approx 0.3 \text{ s}\) and later decreases. Since the jet penetrates the melt bath, slag and melt droplets of various size are separated from the individual phases and the splashing rate considerably increases. The rise in melt level due to the formed depression can also be seen in the figure 8.

![Fig. 8: CFD simulation of a high speed gas jet aligned perpendicular to a slag-melt bath; \(H_l = 1.0 \text{ m}\), \(H_d = 0.26 \text{ m}\), gas injection at \(t_0 = 0 \text{ s}\); other flow conditions according to Tab. 1](image-url)
Fig. 9 shows results for the converter with the six-hole top lance. The instantaneous contour of the slag (blue) and melt (red) surface as well as the velocity vectors in the symmetry plane are presented. The gas phase is treated as incompressible, entering the converter with Ma = 2.21. The transient nature of the blowing process is evident. In particular, just after the start of blow, the rate of slag splashing is high and instabilities in the oxygen-slag and slag-melt interface are observed. The oxygen jets do not interact. However, due to the low lance height, the induced slag and melt depressions overlap and coalesce.

For \( t_1 = 1.3 \) s (Fig. 9a, left column) the jets have not fully penetrated the melt bath (\( H_{\text{cav}} = 0.1 \) to 0.3 m). Although the flow conditions are identical for each nozzle, the shape and depth of the cavities are different. This behavior emphasizes the stochastic character of the process. At that point in time, the slag layer prevents the melt from being splashed and
slag splashing is predominant. Individual slag droplets move upward and get into the supersonic jet zone. Dependent on the mass and momentum of the droplets, the expansion of the jet is disturbed and jet fluctuations are induced. The liquid phases are agitated by the jets and melt rising velocities up to $u_{i,m} \approx 1.0 \text{ m/s}$ in the centre of the converter are generated. The slag flows toward the converter wall with velocities components of $u_{i,sl} \approx 1.0 \text{ m}$. The wear of the refractory lining is expected to be high at the wall-slag interface.

For $t_2 = 2.3 \text{ s}$ (Fig. 9b, right column) the depth and diameter of the depressions increase and the flow pattern becomes more complex. The cavities overlap and interfere. The mean depth is found to be $H_{cav} \approx 0.4 \text{ m}$ (Koria and Lange$^4$: $H_{cav} = 0.68 \text{ m}$). The surface amplitudes of the liquid phases increase and become unstable. The shear stress produced by the high speed gas jet on the slag phases is responsible for the motion and transport of the slag. Due to the low lance height and high nozzle angle, slag droplets are splashed toward the converter wall. In the converter centre, a specific amount of slag and melt is enclosed by the gas jets and splashed onto the lance tip (lance skulling). However, it must be noted that the described flow situation is to be seen as a snapshot of the whole converter blowing process. Especially the desired slag foaming which accelerates the decarburization process due an increase of the specific melt surface can not yet be simulated.

3.3 Mixing and combined blowing

Fig. 10 shows the situation at an arbitrary point in time, when high speed argon gas is released from the tuyeres. The color of the bubble pathlines in Fig. 10a correlates with the rising velocity $u_{i,b}$. The bubble velocity depends on the size, the highest velocity of the large bubbles is $u_{i,b} = 1.6 \text{ m/s}$. The bubbles induce the melt level to be raised about $\Delta H = 0.2 \text{ m}$. The amplitude of the levels is constant. This behavior indicates that the motion of the free surface in combined blowing converters is mainly induced by the top lance. Above the inner tuyeres, the slag layer is displaced by the argon bubbles and so called open eyes appear. However, the bubbles are numerically trapped at the slag-oxygen and melt-oxygen interface.

![Fig. 10: Instantaneous velocity distribution $u_i$ of a) argon bubbles and b) melt during bottom-blowing according to Tab. 3; $t_1 = 46 \text{ s}$, slag-oxygen interface (blue), melt-slag interface (red)](image)

Although the flow rate of each tuyere is the same, the inner arrangement seems to be more efficient. The explanation is found in Fig. 10b. The melt is directed toward the gas plumes, entrained, lifted, spread radially away at the slag-melt interface and is transported downward along the converter wall. The melt flow pattern is characterized by a ring-shaped vortex slightly displacing the outer gas plumes inward. The vortex centre is located at $R = 1.7\,\text{m}$ and $y = 0.75\,\text{m}$ related to the bottom lining. The highest velocity is found adjacent to the bottom ($u_{i,m} \approx 0.3\,\text{m/s}$). In the converter centre, the gas plumes move straight up with a high momentum exchange between bubbles and melt. The melt velocity inside the plumes is $u_{i,m} \approx 1.2\,\text{m/s}$ ($u_{i,\text{slip}} \approx 0.4\,\text{m/s}$). The inner tuyeres form a column of melt with pour mixing, whereas the outer tuyeres induce a melt rollover with intensive mixing.

The influence of the gas plumes on the process of convection and diffusion is described by means of Fig. 11. The concentration field and the pathlines of particles, released from the tuyere position (6) and (9), are shown at an arbitrary point in time. The equilibrium concentration is $c_0 = 2.7 \times 10^{-3}$. The mixing process has not yet been finished. There are still melt regimes which contain a low concentration, especially below the slag-melt interface near the converter side wall. Here, the local velocities are small. However, the point of tracer injection has an influence on the homogenization process.

![Fig. 11: Instantaneous concentration field and pathlines for pure bottom-blowing according to Tab. 3; $t_1 = 46\,\text{s}$ after tracer injection](image)

Fig. 12 shows the mixing efficiency $M$ and the uniformity $U$ for different tuyere configurations A and B. In the latter case, the inner tuyeres are replaced by a centre tuyere. For configuration A, the local maxima of $U$ are found at $t_A = 4\,\text{s}$ and $35\,\text{s}$ ($t_B = 4\,\text{s}$ and $13\,\text{s}$). This behavior is typical for vessels with internal recirculation, see Levenspiel\textsuperscript{11}. The distributions of $M$ and $U$ indicate that the mixing process in case A is more efficient in comparison to case B ($\Theta_{A,95} = 65\,\text{s}$, $\Theta_{B,95} > 80\,\text{s}$). The increase of mixing time with a centre tuyere is also found by Diaz-Cruz et al.\textsuperscript{3} and Olivares et al.\textsuperscript{27}.

The CFD model is able to compare different configurations regarding number, position, flow rate and breakdown of tuyere. In the future, the model will help to clarify some uncertainties concerning the converter melt flow. It is known, i. e., that a certain number of tuyeres at a fixed charge weight and blowing rate must not be exceeded, since exceeding this number
will increase the mixing time. The total melt volume is divided into sub-cells by each gas plume. Since the mixing inside a sub-cell is high, mass and momentum exchange across the cell-borders are low. Increasing the number of tuyeres will increase the number of cell-borders and the mixing process deteriorates. A further increase in number of tuyeres beyond a critical value seems to decrease the mixing time again, because the individual cells overlap.\textsuperscript{17}

Fig. 12: Mixing efficiency $M$ (eq. 15) and uniformity $U$ (eq. 14) for two tuyere arrangements; flow conditions according to Tab. 3

The melt flow pattern is much more complex in the case of combined blowing. According to\textsuperscript{11,17,28}, the interaction between oxygen jets and gas plumes increases the mixing time. As an example, Fig. 13 shows the instantaneous flow field at an arbitrary point in time related to the start of top-blowing.

Fig. 13: Instantaneous volume fraction $\alpha_s$ and velocity $u_i$ distribution for combined blowing according to Tab. 1 and Tab. 3; start of top-blow at $t_0 = 0$ s, start of bottom-blow at $t_3 = 2.3$ s, melt (red), slag (blue)

The gas jets impinge on the bath in close vicinity to the rising gas plumes. Hereby, the depth of the cavities is slightly reduced. A time-dependent, turbulent flow pattern with inten-
sive surface oscillations is induced. On the one hand, the emulsification and foaming of slag is expected to be efficient. On the other hand, the generation of dust by bubble bursting may increase. Surface waves are present throughout the converter process, probably even after the blow has stopped. The gas jets and argon plumes displace the slag layer. The splashing is high compared to pure top-blowing. However, this is a subjective point of view due to a missing numerical quantity to assess the amount of splashing.

4 Conclusion

The BOF process for steelmaking is characterized by a high mass, momentum and energy transfer. Unfortunately, it can not yet be described transparently. For further innovations on the productivity and quality improvement it is necessary to develop numerical approaches. For the process simulation, latest CFD technology has been applied. It was necessary to divide the BOF process in manageable subroutines. The subroutines are analyzed, described and modeled based on different fluid-dynamic observations. Individual solutions are calculated and implemented in the overall model.

The CFD model simulates and predicts the time-dependent, non-isothermal and multiphase effects such as the evaluation of the bath surface and slag/melt splashing. The shape and depth of the cavities induced by oxygen jets are computed. The gas plume induced surface amplitudes can be predicted. Mixing phenomena and mixing time are figured out as well.

The CFD model is used to gain a better insight into the melt flow related phenomena and to make the BOF process more transparent. Further process improvements in terms of efficiency and effectiveness are derived by defining customized solutions for the affecting parameters, i.e. melt bath geometry, number, dimension and arrangement of the bottom blowing and stirring elements, lance height and specific blowing rate. This knowledge supports the design process of new BOF converters. Thus, each converter can be adapted to the individual demands of the customer.

However, CFD in the field of steelmaking is still a challenge. The most significant problem is the simultaneously solution of compressible gas flows in combination with low turbulent, multiphase flows at high densities. In this case, the formulation of turbulence at the fluid interface is not yet satisfactorily.

REFERENCES


