Physical and Mathematical Modeling of the Vessel Oscillation in the AOD Process

Christian WUPPERMANN,1)* Antje RÜCKERT,1) Herbert PFEIFER1) and Hans-Jürgen ODENTHAL2)

1) RWTH Aachen University, Department for Industrial Furnaces and Heat Engineering, Kopernikusstraße 10, Aachen, 52074 Germany. 2) SMS Siemag AG, Eduard-Schloemann-Str. 4, Düsseldorf, 40237 Germany.

(Received on October 15, 2012; accepted on October 31, 2012)

The argon-oxygen-decarburization (AOD) process is a common metallurgical treatment to decarburize high-chromium steel melts using oxygen and inert gas injection through sidewall tuyeres and a top-lance. AOD converters are characterized by a fast and efficient decarburization, whereby the oxidation of chromium is reduced compared to treatments for regular steel grades like the LD process. However, low-frequency oscillations with large amplitudes can occur during the process and influence the converter’s structural integrity. The aim of plant engineering is the development of an AOD converter using a vessel design that provides a fast decarburization rate and effective mixing, whereby the oscillation’s amplitude and the chromium losses are as low as possible. The oscillation of the vessel is induced by the fluid flow. In this study a numerical model is presented, where the oscillation model is integrated in the CFD (computational fluid dynamics) solver by subroutines. The numerical models for both, fluid flow and vessel oscillation, are validated by experiments carried out with a 1:4 scale water model. In a further step, the numerical models are transferred to the actual AOD process. The results of the simulations are compared to experimental results obtained in plant trials. The numerical model developed in the present study can be used as a tool to design AOD vessels that fulfill the above mentioned criteria to satisfy an efficient, reliable and stable process.

KEY WORDS: AOD process; vessel oscillation; water modeling; CFD; plant trials.

1. Introduction

During the last four decades the AOD process has been established as a reliable and efficient refining process in stainless steelmaking. Compared to other injection geometries, e.g. bottom blowing, the injection of process gases (oxygen, nitrogen, argon) through sidewall tuyeres shows advantages in mixing effectiveness.1) The steel grades, which are refined in AOD converters, contain usually more than 10% of chromium. Figure 1 shows a typical blowing procedure for the 120 ton AOD converter investigated in this study. The composition of the process gas and the oscillation amplitude were determined in plant trials prior to this investigation. The composition of the process gas in the first period of the refining process (cf. Main Blow in Fig. 1) contains about 90% oxygen, which is used as decarburization agent. At the decarburization stage elements with the highest affinity to oxygen (carbon, silicon, and manganese) start to oxidize at first. With decreasing amount of those elements the tendency of chromium oxidation increases. That is why the amount of inert gases (nitrogen and argon) is increased with proceeding process time. Thus, the partial pressure of carbon monoxide (pCO) in the gas phase is reduced, and the reaction equilibrium is forced towards the oxidation of carbon. The reaction and the corresponding equilibrium constant K describing the Cr-C-equilibrium are shown in Eqs. (1) and (2).

\[ 2[Cr] + 3[CO] \leftrightarrow 3[C] + (Cr_2O_3) \] (1)
The first important criterion which must be considered during the design of the AOD vessel and the surrounding construction is the mixing behavior, in order to increase the efficiency of the desired refining reactions. The second criterion is that a stable process without disturbances or inappropriate operating conditions must be ensured. An example of inappropriate operating conditions during the AOD process is the occurrence of vessel oscillations. The oscillation behavior of the AOD vessel is basically independent of the mixing behavior which was investigated in a previous study. A proper design of an AOD converter has to fulfill both criteria: Fast mixing and low oscillation amplitudes. The investigation of the latter phenomenon is the focus of the present study.

The oscillations with the largest amplitudes occur in process stages with the highest amount of injected inert gases as shown in Fig. 1. This tendency is also described elsewhere. The data shown in Fig. 1 was carried out in plant trials on a 120 ton AOD vessel.

During the dynamic blow stages the slag is in a granular state and the injected process gas is a mixture of inert gases and oxygen. The computational effort to simulate a granular slag layer above a liquid steel melt as well as the consideration of chemical reactions at the metal-gas interface (CO-formation) is high. The uncertainty of the values used for the physical properties such as the diameter distribution of the slag chunks and their shape factor regarding a granular slag is high. Hence, the reduction stage has been simulated to simplify the numerical model. In this period the slag is assumed to be liquid and the gas injection of pure argon is performed through tuyeres without the top-lance. The gas jets enter the system at a velocity according to a Mach number of approximately Ma = 1. The initial momentum of the gas jets is absorbed within a short distance from the tuyere exit, due to the large density difference between steel and gas of approximately 1:500 (assuming an tuyere outlet density of \( \rho_{\text{Ar}} \approx 15 \text{ kg/m}^3 \)). The buoyancy force acting on the gas phase changes the jet’s trajectory into a vertical bubble column (plume). The drag force between gas and melt leads to the characteristic AOD flow pattern shown in Fig. 2 and described by various authors. Thus, the stirring energy of the argon bubbles, which is predominant during the AOD reduction process stage, is introduced into the melt. The transfer of forces between fluid and vessel is caused by the shear and pressure forces acting on the walls. The sloshing motion of the free surface causes oscillating pressure forces. The viscous forces perpendicular to the wall area are caused by the velocity boundary layer. The mechanism of energy transfer starting from the injected gas jets over the fluid motion to the vessel oscillation is sketched in Fig. 3.

The parameters, that influence oscillation behavior, are the vessel’s design, the melt capacity, the thickness of the slag layer, the physical properties of the slag, the composition and the volume flow rate of the process gases, the tuyere number and their arrangement. Increased stirring leads to shorter mixing times, more efficient slag melting during the reduction stage and an increase in productivity, but also to higher vessel oscillations.

Kojima et al. investigated the oscillation behavior of a combined blowing converter and the corresponding 1:10 scale water model. The analysis of the frequency spectra shows two superimposed frequencies. The lower frequency peak corresponds to the sloshing motion of the liquid surface. This frequency peak can be determined using an analytical expression derived by Abramson. Another semi-empirical equation describing the motion of the free
surface in the AOD process has been published by Xie et al.\textsuperscript{12} The higher frequency peak corresponds to the mechanical eigenfrequency of the vessel system, which can be determined by setting up and solving the differential equation of the vessel’s motion. Fabritius et al.\textsuperscript{4} measured the vessel oscillation of a 150 ton AOD converter and a water model. They compare the measured frequencies to calculated ones. The calculations are based on equations describing the sloshing motion of the free surface derived by Abramson\textsuperscript{11} and Xie.\textsuperscript{12} The agreement between the frequency peaks observed during the water model tests and those calculated using the analytical method is very good. In contrast, there is a large discrepancy between the analytically calculated and the measured frequency peaks for the 150 ton AOD vessel. Fabritius does not consider the existence of both frequency peaks. He neglects the characteristic frequency of the vessel system. This fact explains the discrepancy between the calculated and the measured frequency peak for the 150 ton AOD vessel.

There are several studies published that deal with the numerical simulation of the fluid flow in AOD vessels\textsuperscript{7,13–17} but the transfer between CFD and the simulation of the oscillating structure is not carried out.

This transfer between CFD and the oscillation of the structure is the focus of the presented study. The oscillation is primarily a feature of the vessel’s structure, but the driving force for the motion is the fluid flow in the vessel. Therefore, a coupling of CFD and FEM (finite element method) solvers has to be carried out to simulate the vessel motion numerically. An oscillation model has been developed that is integrated in the CFD solver as a sub-routine, to reduce the computational effort. Experimental tests in a 1:4 scale water model of a 120 ton AOD converter are performed to validate the model. In a further step the model is applied on the 120 ton AOD process. The results of these simulations are compared to experimental results carried out in plant trials.

2. Physical Simulation

2.1. AOD Converter

Figure 4 shows a schematic of the 120 ton AOD vessel. After the lifetime of the refractory lining the vessel is lifted out of the trunnion ring and replaced by a vessel with new lining. The process gases are injected into the melt by seven sidewall tuyeres. The tuyeres are arranged in a horizontal plane and are adjusted radially towards the centerline at an angle of $\alpha_t = 18^\circ$ between each tuyere, see Fig. 4. The pivotable parts are the actual vessel, the trunnion ring and the shafts which are connected to the frame over bearings. A gear box is used to turn the vessel for charging and discharging purposes. The mechanical load during the process is compensated by a torsional shaft acting like a spring that forces the vessel and the gear box back into their resting position. To determine the mechanical boundary conditions (spring constant $c_{AOD}$, damping constant $d_{AOD}$, mass moment of inertia $\theta_{AOD}$), a computer-aided-design model (CAD) has been set up and evaluated mechanically. The geometric data and the mechanical boundary conditions are summarized in Table 1.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner diameter (cylinder) $D_{AOD}$</td>
<td>$\sim 3$</td>
<td>m</td>
</tr>
<tr>
<td>Steel level (design point) $H_s$</td>
<td>2.096</td>
<td>m</td>
</tr>
<tr>
<td>Slag layer $H_{sl}$</td>
<td>$\sim 0.3$</td>
<td>m</td>
</tr>
<tr>
<td>Diameter of the sidewall tuyeres $D_t$</td>
<td>0.016</td>
<td>m</td>
</tr>
<tr>
<td>Height of the tuyeres $h_t$</td>
<td>0.5</td>
<td>m</td>
</tr>
<tr>
<td>Angle between sidewall tuyeres $\alpha_t$</td>
<td>18</td>
<td>degree</td>
</tr>
<tr>
<td>The vessel’s moment of inertia (all pivotable parts) $\theta_{AOD}$</td>
<td>constant</td>
<td>kg m$^2$</td>
</tr>
<tr>
<td>Spring constant $c_{AOD}$</td>
<td>constant</td>
<td>Nm/ rad</td>
</tr>
<tr>
<td>Damping constant $d_{AOD}$</td>
<td>0</td>
<td>(Nm s)/rad</td>
</tr>
<tr>
<td>Center of rotation $h_a$</td>
<td>$\sim 3$</td>
<td>m</td>
</tr>
<tr>
<td>Steel density $\rho_s$</td>
<td>7033</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>Slag density $\rho_d$</td>
<td>2990</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>Gas density (STP) $\rho_g$</td>
<td>1.784</td>
<td>kg/m$^3$</td>
</tr>
</tbody>
</table>

2.2. Water Model Setup

Figure 5 shows the experimental setup used for the determination oscillation frequencies in the water model. The 1:4 scale vessel consists of acrylic glass and is geometrically

![Fig. 4. Schematic of the 120 ton AOD converter with gear box and torsional shaft.](image1)

![Fig. 5. Drawing of the 1:4 scale water model.](image2)
similar to the 120 ton AOD converter. The tuyeres are arranged in a horizontal plane and are adjusted radially towards the centerline at an angle of \( \alpha_t = 18^\circ \), see Fig. 6. A compressor followed by a pressure-reducing valve provides a constant volume flow rate of the air. The volume flow rate can be controlled by a flow meter, whereby each tuyere is equipped with a metering valve. The pivotable parts of the water model are the acrylic glass vessel, the trunnion ring and the tilting axis. These parts are connected to the frame over two bearings on each side of the vessel. A torque support arm, which is connected between the tilting axis and the frame, acts like a mechanical spring that forces the vessel to rotate back in its resting position if a deflection is induced.

The oscillation is monitored by a displacement sensor and a strain gauge which quantify the vessel’s rotation around the tilting axis (x-axis). The displacement sensor is located at the bottom of the vessel (y = 0). The tilting angle (\( \phi \)) can be calculated using the vertical distance between tilting axis and vessel bottom (\( h_a \)) and the measured displacement. The strain gauge is attached to the torque support arm, which is bent when the vessel rotates around the tilting axis. A calibration procedure has been carried out to obtain the relation between the tilting angle (\( \phi \)), the torque acting on the torque support arm and the corresponding current of the strain gauge. The spring constant \( c \) and the damping constant \( d \) have been determined experimentally. A data logging system is used to monitor the signals of both sensors versus time. The data is post-processed using statistical methods and Fast-Fourier-Transformation (FFT). The geometric data and the mechanical boundary conditions are summarized in Table 2.

### 2.3. Similarity Considerations

The movement of the gas plume in the AOD vessel depends on the equilibrium of buoyancy, gravity, and inertia forces acting on the fluids which are simulated. The jet injection and the trajectory of the plume are characterized by the modified Froude number, which is commonly used in literature as criterion to reach kinematic similarity between AOD converters, prototypes and water models.\(^{18,19} \)

\[
Fr_m = \frac{\rho_g u_t^2}{\rho_l g D_t} \quad \text{(for } \rho_g \ll \rho_l \text{)} \quad \text{ (3)}
\]

- \( \rho_g \): Gas density at the tuyere exit in kg/m\(^3\)
- \( \rho_l \): Liquid density in kg/m\(^3\)
- \( g \): Gravity, 9.81 m/s\(^2\)
- \( u_t \): Mean gas velocity at the tuyere exit in m/s
- \( D_t \): Tuyere diameter in m

Gas density and velocity at the tuyere exit have to be calculated in terms of gas dynamics, as the flow in the tuyere exceeds Ma > 0.3.\(^{20} \)

### 3. Numerical Model

#### 3.1. CFD Model

The solution of the RANS-equations (Reynolds-Averaged-Navier-Stokes) for the fluid flow in the AOD vessel is performed with ANSYS FLUENT 14. A transient formulation of the RANS-equations is chosen in order to resolve oscillating flow features, which force the vessel to oscillate. The turbulence properties of the continuous phases are calculated using the Scale-Adaptive Simulation Model (SAS model) described by Menter \( et \) \( al. \)\(^{21,22} \). This model shows advantages in the resolution of turbulence phenomena compared to conventional two equation models (k-\( \omega \) or k-\( \varepsilon \)).\(^{23} \) The pressure forces acting on the vessel wall play a major role in the agitation of the system. Thus, a multiphase model has to be used to predict a realistic shape of the free surface and produces a sharp interphase between the involved fluids simultaneously. The VoF-model (Volume of Fluid model) according to Hirt and Nichols\(^{24,25} \) fulfills both criteria.

The injected process gases introduce the desired stirring energy into the system to agitate the fluid flow. The injected gas (argon) is not modeled as an Eulerian phase. On the one hand the initial momentum of the gas jets is very large. The interaction of the compressible gas phase with the surrounding liquid, with its very large density ratio, leads to problems of stability in the CFD-solver used for the present study. On the other hand, the use of an explicit Eulerian

---

**Table 2. Geometry and process data of the water model.**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner diameter (cylinder)</td>
<td>D</td>
<td>0.931 m</td>
</tr>
<tr>
<td>Fluid level (design point)</td>
<td>( H_f )</td>
<td>0.524 m</td>
</tr>
<tr>
<td>Diameter of the sidewall tuyeres</td>
<td>( D_t )</td>
<td>0.002 m</td>
</tr>
<tr>
<td>Height of the tuyeres</td>
<td>( h_t )</td>
<td>0.112 m</td>
</tr>
<tr>
<td>Angle between sidewall tuyeres</td>
<td>( \alpha_t )</td>
<td>18 degree</td>
</tr>
<tr>
<td>The vessel’s moment of inertia (all pivotable parts)</td>
<td>( \theta )</td>
<td>112 kg m(^2)</td>
</tr>
<tr>
<td>Spring constant</td>
<td>( c )</td>
<td>6.062 Nm/rad</td>
</tr>
<tr>
<td>Damping constant</td>
<td>( d )</td>
<td>0.1 (Nm s)/rad</td>
</tr>
<tr>
<td>Center of rotation</td>
<td>( h_a )</td>
<td>0.719 m</td>
</tr>
<tr>
<td>Liquid density</td>
<td>( \rho_l )</td>
<td>998 kg/m(^3)</td>
</tr>
<tr>
<td>Gas density (STP)</td>
<td>( \rho_g )</td>
<td>1.293 kg/m(^3)</td>
</tr>
</tbody>
</table>
multiphase model, like the VoF-model, would require a very fine computational grid in the area of the gas plume in order to resolve the phase boundary between the fluid and the gas phase. Hence, the process gas is simulated by a Lagrangian approach, to reduce the computational effort and to reach a stable calculation of the fluid flow. The bubbles are treated as discrete particles with constant density. The model is called discrete phase model (DPM). The interaction between the discrete and the continuous phase is taken into account. Drag and buoyancy forces are used to calculate the particles’ acceleration subsequently according to Eq. (4): \[ \frac{d \mathbf{u}_p}{dt} = F_D \left( \mathbf{u} - \mathbf{u}_p \right) + \frac{\mathbf{g} \cdot (\rho - \rho_p)}{\rho_p} \] with: \[ F_D = \frac{3 \cdot \mu \cdot C_D \cdot Re}{4 \cdot \rho_p \cdot d_p^2} \] \[ C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \] \[ u_p \] Particle velocity in m/s \[ t \] Time in s \[ u \] Velocity of the continuous phase in m/s \[ g \] Gravity vector in m/s² \[ \rho_p \] Particle density in kg/m³ \[ \rho \] Density in the continuous phase in kg/m³ \[ C_D \] Drag coefficient \[ \mu \] Molecular viscosity of the continuous phase in kg/(m² s) \[ a_{1,2,3} \] Drag law coefficients \[ d_p \] Particle diameter in m \[ Re \] Particle Reynolds number

The particles’ effect on the flow field and the turbulence of the continuous phase and vice versa is simulated using a two-way turbulence coupling between the both phases.

The bubble size distribution for the air/water system is shown in Eq. (7).

\[ \phi_{\text{in}} = \sum_{i} M_i \theta \cdot \phi = \sum_{i} M_i \] \[ \Leftrightarrow \theta \cdot \phi = \sum_{n} \left( \phi_{n-1} + \phi_n \right) \cdot \Delta t \]

Equation (6) has been derived assuming ideal gas law as the equation of state and isothermal conditions. The particles are removed from the computational domain when they reach the phase boundary between liquid and gas phase. Numerically spoken the particles are deleted in cells where the volume fraction of gas exceeds vofgas > 0.5. Table 3 provides an overview of the chosen solver settings for the fluid flow computations.

### 3.2. Oscillation Model
The trajectory of a body is described by six degrees of freedom (three rotational and three translational degrees). The motion with the largest amplitude of the object of investigation during the process is the rotation around the tilting axis. The complexity of the motion is reduced to one degree of freedom in order to describe the above mentioned rotation. This simplification allows describing the vessel oscillation as the motion of a mass pendulum shown in Fig. 7. The vessel’s and trunnion ring’s structures are assumed to be rigid bodies. The free rotation of the vessel is inhibited by the torque support arm in the water model and the torsional shaft for the 120 ton AOD converter respectively. The pendulum’s rotation can mathematically be described by the principle of conservation of the angular momentum as shown in Eq. (7).

\[ \theta \cdot \phi = \sum_{i} M_i \]
MF(t) Fluid torque acting around the tilting axis in Nm
θ Mass moment of inertia in kg m²
c Spring constant in Nm/rad
d Damping constant in Nm s/rad
g Gravity, 9.81 m/s²
m Mass of the vessel in kg
l Distance between the centers of gravity and rotation in m
ϕ Tilting angle in rad
ϕ Rotational velocity in rad/s
ϕ Rotation acceleration in rad/s²
n Current time step
n-1 Previous time step
Δt Time step size in s

The motion of the fluid inside the vessel causes pressure and shear forces acting on the wall of the vessel. The non-symmetric distribution of those forces causes a torque around the tilting axis which agitates the oscillation of the vessel. The torque is described by MF(t) and is calculated by a UDF integrated in the CFD solver. In a first step the pressure and shear forces on each wall cell \( \overrightarrow{F}_{pi,i} \) and \( \overrightarrow{F}_{i,i} \) are calculated. Then, the resulting torque around the tilting axis for each wall cell is calculated as shown in Eq. (8) and Fig. 8.

\[
\tau \text{FM} = \sum_i \tau \text{Mi,i}
\]

and the resulting torque is calculated as shown in Eq. (8) and Fig. 8. The integrated torque is one of the input variables in Eq. (7). By means of the constants c, d, \( \theta \) and the time dependent torque \( MF \) the angular acceleration \( \phi \) can be calculated. The other rotational coordinates \( \varphi \) and \( \phi \) are then determined using an explicit scheme as shown in Eq. (7). The rotation of the mesh is performed by the “moving mesh function” in ANSYS FLUENT 14. The input variables are the rotational velocity \( \phi \), the tilting axis’ position and the tilting axis’ direction. After the rotation of the mesh, the flow and turbulence field for the subsequent time step is calculated by the CFD solver and the cycle starts again. Figure 9 illustrates the numerical procedure of the simplified oscillation model for one time
In addition the model is capable of describing more than one degree of freedom. It can be expanded if necessary. Besides the rotational motion translational modes can be simulated as well.

4. Results and Discussion

4.1. Validation of the Numerical Models by the Water Model Experiments

The model is validated by measurements carried out with the water model. The water model has one major advantage compared to measurements at the actual AOD converter. The experiments are simple to execute and parameter variations can be carried out easily. The parameter with the largest influence on the oscillation frequency is the liquid fill level. Thus, a variation of the fill level is performed to validate the numerical model. In Table 4 the sensitivity analysis sequence for the numerical simulations and the experiments are presented. The Hf/D-ratio is varied between 0.42 < Hf/D < 0.79. Due to the complexity of the water model it is not possible to change the diameter D. The Hf/D-ratio can only be varied by changing the fill level Hf. With a carefully validated numerical model it is possible to characterize the influence of the vessel’s diameter on the oscillation behavior. This still has to be confirmed in further investigations. Nine experiments and four numerical simulations are carried out. Figure 10 shows the comparison of the typical frequency spectra obtained in the water model tests and the numerical simulations. Regarding the numerical simulation the frequency spectra of the agitation of the vessel oscillation Mf and the system response ϕ are shown. All spectra indicate that the vessel oscillation consists of two superimposed frequencies represented by two dominating peaks. The first peak (f1 ≈ 0.9 Hz) corresponds to the sloshing motion of the free surface. The assignment can be proven by analytical equations for lateral waves in cylindrical tanks derived by Abramson, shown in Eq. (9).

\[ f_1 = \frac{\sqrt{K \cdot g}}{2 \cdot \pi^2 \cdot D} \cdot \tanh \left( \frac{2 \cdot K \cdot H_f}{D} \right) \]  

(9)

f1 Frequency of the sloshing motion of the free surface in Hz
K Geometry specific constant

The second frequency (f2 ≈ 1.3 Hz) is in the characteristic frequency of the vessel system which is the solution of the homogeneous part of the differential Eq. (7), shown in Eq. (10):

\[ f_2 = \frac{1}{2\pi} \sqrt{\frac{c}{\theta}} \]  

(10)

f2 Characteristic frequency of the vessel system in Hz

The agreement between the frequency spectra obtained in the numerical and physical simulations is excellent. On comparing the numerical results (Fig. 10) an augmentation of the intensity of frequency f2 related to f1 can be identified between the agitation of the system (Mf) and the system response (ϕ). This indicates a resonance phenomenon that is suspected to cause vessel oscillations with large amplitudes during the AOD process.

The validation of the numerical model is performed in two steps. In the first step, the eigenfrequencies of the vessel are measured and simulated without the influence of the injected process gas. The vessel is tilted by a specific angle ϕinit. After the vessel is released it conducts a damped oscillation around its resting position due to gravitation and inertia forces. During the described oscillation the system is not disturbed by any random agitation. Thus, the eigenfrequencies of the vessel can be determined and their dependency on changes in the Hf/D-ratio can be analyzed. The model is

![Fig. 10](attachment:image.png)

**Table 4.** Parameters and measurement program for the validation of the numerical model.

<table>
<thead>
<tr>
<th>Fill level in % (referred to the design point)</th>
<th>Hf/D</th>
<th>Num.</th>
<th>Exp.</th>
</tr>
</thead>
<tbody>
<tr>
<td>70</td>
<td>0.42</td>
<td>–</td>
<td>yes</td>
</tr>
<tr>
<td>80</td>
<td>0.47</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>90</td>
<td>0.52</td>
<td>–</td>
<td>yes</td>
</tr>
<tr>
<td>100</td>
<td>0.56</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>110</td>
<td>0.61</td>
<td>–</td>
<td>yes</td>
</tr>
<tr>
<td>120</td>
<td>0.66</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>130</td>
<td>0.70</td>
<td>–</td>
<td>yes</td>
</tr>
<tr>
<td>140</td>
<td>0.75</td>
<td>yes</td>
<td>yes</td>
</tr>
<tr>
<td>150</td>
<td>0.79</td>
<td>–</td>
<td>yes</td>
</tr>
</tbody>
</table>
tilted by the initial angle of \( \phi_{\text{init}} = 0.5^\circ \) around the tilting axis. The angle \( \phi_{\text{init}} \) is chosen because it is in the same range as the typical oscillation amplitude obtained in the water model tests with gas injection. Figure 11 shows the behavior of the two frequencies with respect to the fill level. For \( f_1 \), the agreement between numerical simulation, experiment and analytical solution is very good. With increasing \( H_f/D \)-ratio the frequency \( f_1 \) also increases. The numerical simulation seems to over-predict the frequency \( f_2 \) by 4.5% (average), but the agreement between numerical simulation and experiment is still very good and the decreasing tendency with increasing \( H_f/D \)-ratio is predicted well.

In the second step of the validation the characteristic frequencies are measured and simulated with injection of the process gas. Figure 12 shows the characteristic frequencies with respect to the \( H_f/D \)-ratio. Regarding \( f_1 \) the agreement between the water model tests and numerical simulation is very good. The averaged variance is 3.7%. The agreement for \( f_2 \) is inferior compared to \( f_1 \) especially for \( H_f/D \)-ratios lower than 0.65. In those regions the numerical model under-predicts the frequencies obtained in the water model. But with an averaged variance of 6.8% the agreement is still sufficient.

### 4.2. Transfer of the Numerical Models to the Actual AOD Process and Validation by Experimental Results from Plant Trials

The accuracy of the oscillation model has been tested in the presented water model study. In the next step the model is transferred to the actual AOD process. The geometry of the numerical mesh is developed on the basis of the inner shape of an AOD vessel with new lining. The plant trials were also performed in periods with a renewed refractory lining. The geometric data, the process data, the numerical boundary conditions and the utilized models are shown in Tables 1 and 3. As shown in section 3, a transient formulation is used to simulate the flow field. Due to the fact that the presentation of the transient flow field is a complex issue, the time-averaged flow field is illustrated in Figure 13, in order to get a general impression of the flow pattern. A time interval of 20 s is used for the calculation of the averaged variables. The total volume flow rate through all tuyeres is set to \( \dot{V}_{\text{Ar(STP)}} = 120 \text{Nm}^3/\text{min} \) and the fill level is set to \( H_f/D = 0.56 \). Both parameters represent the design point conditions of the 120 ton AOD vessel. All plant trials are performed using these parameters.

The measurements of the vessel oscillations are carried out with strain gauges attached to the retaining rods which connect the gear box to the torsionalshaft (Fig. 4). The sampling rate is 100 Hz. The ratio of \( f/f_{\text{max}} \) presented in a dimensionless form is taken to understand the oscillation behavior. Therefore the obtained frequencies are divided by the constant \( f_{\text{max}} \).

Figure 14 shows the simulated and the measured frequency spectra. The frequency of the free surface’s sloshing motion is located at \( f/f_{\text{max}} = 0.1 \) and the characteristic frequency of the vessel system at \( f/f_{\text{max}} = 0.51 \). The measured frequency \( f_2 \) is approximately 1.9% higher compared to the simulated one. The deviation concerning the frequency \( f_1 \) is much higher. The value for \( f_1 \) obtained in plant trials is 14% lower compared to the simulated one. The uncertainty of the assumed viscosity of the liquid slag could cause the large deviation concerning \( f_1 \). With increasing viscosity the damping property of the slag acting on the free surface motion is also increased, see. A larger damping leads to lower frequencies. A detailed investigation on material properties of the slag and its influence on the oscillation behavior of the AOD vessel is to be addressed in future work.

The agreement between numerical simulation and plant trials for frequency \( f_1 \) is sufficient and for \( f_2 \) very good. This fact shows that the oscillation model is also capable of predicting the frequencies for the vessel oscillation in the 120 ton AOD process.
5. Conclusions

The oscillation frequencies of a 120 ton AOD converter and a 1:4 scale water model have been measured and analyzed. A numerical model coupling the fluid dynamics and the vessel oscillation has been developed and successfully validated. The measured and simulated frequency spectra show very good agreement between the numerical simulation and the experimental data.

- The measured and simulated frequency spectra show that the vessel oscillation is dominated by two superimposed frequencies. The lower one represents the sloshing motion of the free surface and the higher one the characteristic frequency of the vessel system.
- The numerical simulation indicates that the agitation of the system (M1) causes a resonance in the system response (\(\phi\)). An augmentation of the intensity of the second frequency peak \(f_2\) in the system response spectrum in comparison to its intensity in the agitation spectrum can be identified.
- The validation shows very good agreement between the simulated frequency spectra and those which are obtained in water model tests.
- The comparison to results gained in plant trials proves that the model is capable to predict the oscillation frequencies of an AOD vessel during process.

- The oscillation model can be easily expanded to more than one degree of freedom. Rotational degrees as well as translational degrees can be added, if necessary.
- The influence of the vessel’s diameter on the oscillation behavior can be analyzed by numerical simulations. This investigation is to be addressed in future work.
- The numerical model is used to design the vessel shape and other structural elements of AOD converters in order to avoid resonance and to ensure a reliable and stable process.
- To design an efficient AOD process the presented oscillation model can be used in combination with results of previous studies concerning mixing effectiveness.25

REFERENCES