EXPERIMENTAL AND NUMERICAL INVESTIGATIONS OF A TWIN BELT CASTER ON THE GROUND OF A WATER MODEL AND SIMULATIONS

Nils Lichtenberg1, Philipp Kinzel1, Nicole A. Parks1, Dominique Thévenin1, Ulrich Urlau2

1Laboratory of Fluid Dynamics and Technical Flows (LSS) University of Magdeburg “Otto von Guericke” Universitätsplatz 2, 39106 Magdeburg, Germany E-mail: Nils.Lichtenberg@ovgu.de

2Urlau Innomanagement GmbH Oberhusrain 47, 6010 Kriens, Switzerland E-mail: info@urlau-innomanagement.com

ABSTRACT

This article describes a detailed investigation of the flow at the entry of a twin-belt caster, combining non-intrusive flow velocity measurements via Particle-Image Velocimetry (PIV) with detailed numerical simulations based on Computational Fluid Dynamics (CFD). The experimental set up includes a tundish, a stopper, a Submerged Entry Nozzle (SEN), and the walls of the twin-belt caster. Similarity theory was used to build an appropriate water model. The water level in between the horizontal belts is controlled by the height of the fully functional stopper. Detailed experimental measurements deliver the velocity distribution at the outlet of the SEN and are used to characterize the jet between the belts. The simulation identically reproduces this model providing a direct comparison. Two different turbulence models are applied in CFD simulations. The differences found between the results provide interesting insights. The k-ɷ-SST turbulence model delivers a much better agreement and is thus recommended for future studies. It is also found that, due to the very complex flow structures appearing around the stopper head, the full geometry must be considered in the simulation. Slow fluctuations are observed both in the course of the experiments and CFD, which is why meaningful comparisons are only possible for time-averaged quantities. Finally, a very good agreement is obtained between the measurements and the simulations, opening the door to further studies regarding caster flow homogeneity.

Keywords: Belt caster, Computational Fluid Dynamics (CFD), turbulence model, Particle Image Velocimetry (PIV), water model, tundish, stopper, LUUP, Compact Strip Production (CSP).

INTRODUCTION

With its almost 3000-year-old history, the steel industry certainly qualifies as mature. Nevertheless, it is expected to continue to grow at an average rate of 2.9% over the next decade [1]. Steel is characterized by intense competition, which, above all, drives increases in quality and decreases in manufacturing costs. During the past four decades approximately 95% of “thinner” long steel products have been produced via the conventional continuous casting process followed by hot forming. A substantial, disruptive improvement of this process route would therefore seem unlikely. However, the process idea known as LUUP represents such a new production route. LUUP is an acronym that stands for:

- **LUUP**
  - **L**ong products production process, characterized by
  - **U**ltra low investment and production costs
  - **U**ltra low environmental impact
  - **P**roduct properties at the top level.

LUUP is based on thin-slab casting methods such as the CSP (Compact Strip Production) process and the Contirod technology found in copper wire production [2]. Compared to the state of the art, it has the potential to improve significantly the product quality, while lowering substantially the production costs [3].

In essence, the process consists of a twin-belt caster and a subsequent in-line hot-forming step (Fig. 1).

The technological, economic, ecological, and quality aspects of the LUUP process will not be addressed here.
Instead, our work focuses on the central element of this process, the so-called twin-belt caster (Fig. 2).

In the twin-belt caster, liquid steel is poured between the surrounding walls. In contrast to the conventional continuous casting, the steel is poured into molds with oscillating side walls. In the course of a single oscillating cycle the wall precedes or moves against the main casting direction.

The first strand shell is produced within the twin-belt caster. Due to the high temperatures and the low thickness, it is very soft. Therefore, optimal solidification conditions are required to minimize the induced stresses. The flow behavior plays a key role here. It is thus necessary to derive predictive models describing the impact of the flow on solidification; with such models, one can then manipulate the flow to reach optimal conditions. As a first step toward a predictive model of the flow within the twin-belt caster, a validation step is necessary.

CFD (Computational Fluid Dynamics) can provide insights into the otherwise difficult-to-access inner workings of the casting plant. It allows for a detailed study of the liquid metal and its velocity fields, pressure distribution, and other physical values needed for the flow characterization.

We have so far studied a belt caster at the Laboratory for Fluid Dynamics and Technical Flows (LSS) using extensive numerical and experimental methods. The special features of a belt caster are the previously-mentioned moving walls (belts), which represent the counterpart to the mold in the conventional continuous casting. Because the walls move at the casting speed of the steel, a more uniform velocity field can be achieved. This, in turn, leads to lower stresses within the metal structure during steel solidification.

Water models represent one way to gather experimentally information on the belt casting process. However, testing all of the desired configurations is hard to realize within the water model itself, due to the spatial restrictions and the high costs. As a complementary solution, CFD simulations can be used.

Compared to the experiments using a water model, the numerical simulations provide the easy variation or combination of the different parameters. These may include:

---

Fig. 1. LUUP principle sketch - inline casting and rolling.

Fig. 2. A ladle, connected to the tundish by a ladle shroud. The stopper height regulates the outflow through a SEN into the belt casting system consisting of rotating belts and two dams.
- the production rate per caster;
- the caster geometry (width, height, length);
- the caster orientation;
- the submerged nozzle geometry;
- the nozzle submersion depth;
- the connection between the tundish and the caster.

Since CFD simulations of turbulent flows deliver only an approximation of the truth, one must first compare the CFD results with experimental measurements for a realistic configuration, that is, one must validate the numerical model.

For the process considered here, the inlet conditions into the belt caster are of central interest. As a consequence, the validation considers the velocity field at the outlet of the SEN (submerged entry nozzle), where the flow enters the belt caster. This particular location is investigated both experimentally, using particle image velocimetry (PIV), and numerically by CFD. Since it is known from the scientific literature that the employed turbulence models impact strongly the CFD predictions, results from two different models will be compared.

**EXPERIMENTAL**

**Water Model**

The water model was used to mimic the supply of the liquid steel through an existing continuous casting plant with a tundish, a stopper, and a ladle shroud.

The real, industrial system is illustrated in Fig. 2. It comprised a ladle with liquid steel. The tundish was positioned underneath the ladle connected by a ladle shroud. The stopper was located within the tundish and had an adjustable height, regulating thus the outflow of the liquid metal through the SEN into the twin-belt caster complex. The belt caster consisted of two rotating belts, which were closed off at their sides by two rotating dams to prevent the liquid steel escape [4]. The partially solidified steel billet then emerged at the end of the strip caster and had a cross section of 175 mm x 125 mm. The dimensions were comparable to those of the conventional billets used in the continuous casting process.

The water model was specifically derived to represent the industrial process. All structural components were constructed true-to-scale using Plexiglas (allowing for optical measurements) or epoxy resin, and mounted on a 3 m tall pedestal. The volume flow rate was identical to that of the real process. The similarity theory [5] was systematically used relying on the fact that the liquid steel at the process temperature and the tap water regulated at a temperature of 30°C had nearly the same kinematic viscosity (Table 1). Using this approach, the key nondimensional numbers characterizing this flow (Reynolds number and Euler number) were identical in the real process and in the water model.

Since an optical access was required for PIV measurements, not all structural components of the real strip casting plant were represented in the water model. In particular, fixed optically-transparent Plexiglas walls replaced the (moving) belts and the dams of the casting plant in the water model. The ladle, being irrelevant for the flow at the outlet of the SEN, was also neglected; only the ladle shroud was kept as shown in Fig. 3. As in the real process, the flow from the tundish into the belt caster was regulated by the stopper height. The stopper geometry was kept unchanged. The behavior of the fluid system was therefore more complex than in most of the similar water models like that documented in ref. [8], where it had no regulating function. In order to get the target process conditions of 12.5 m³ water/h exiting the tundish into the belt caster, the stopper was finally lifted to a constant height of 3.65 mm during the water model operation.

**PIV Measurements**

The first area of interest was located in the caster, just after the SEN. Accurate PIV measurements required

<table>
<thead>
<tr>
<th>Process property</th>
<th>Steel [°C]</th>
<th>Water [°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>1600</td>
<td>30</td>
</tr>
<tr>
<td>Kin. viscosity</td>
<td>0.852 [6]</td>
<td>0.801 [7]</td>
</tr>
<tr>
<td>Density</td>
<td>7200</td>
<td>1000</td>
</tr>
<tr>
<td>Mass flow rate</td>
<td>90000</td>
<td>12500</td>
</tr>
<tr>
<td>Volume flow rate</td>
<td>12.5</td>
<td>12.5</td>
</tr>
</tbody>
</table>

Table 1. Material properties relevant for similarity theory calculations.
first a calibration. For this purpose, a calibration target was positioned at the location of the later measurement planes within the water flow, as illustrated in Fig. 4, and imaged by CCD camera (Dantec CV-M2).

A double-pulse 200 mJ Nd:YAG “laser” (Co. New Wave) with a wavelength of 532 nm and a pulse frequency of 15 Hz was used. The total recording time was 10 s, thereby providing 150 double-images that were afterwards averaged to obtain the resulting velocity field. With a standard planar PIV as used here, the velocity vectors were obtained over the entire cross-section illuminated by the “laser” light. This was represented by the area of the calibration target shown in Fig. 4. However, only planar information was thus obtained; the third velocity component (pointing out of the “laser” sheet) could not be determined in this manner.

In order to still get some information concerning the possible three-dimensional flow features, both a vertical measurement plane (as the calibration target in Fig. 4), and a perpendicular, horizontal plane were investigated by PIV.

**NUMERICAL INVESTIGATIONS**

Aiming a comparison with the results of the water model investigations, a CFD model (“virtual tundish”) was developed. It kept geometry and dimensions identical with those of the water model. A simplified domain starting with the inflow of the SEN but leaving the tundish out (region 1 in Fig. 3) was initially considered to decrease the computational time. As explained later, a satisfactory comparison with PIV results could not be achieved with this short computational domain. Therefore, at the expense of considerably higher computational requirements, the full geometry (region 1 plus region 2 in Fig. 3) was finally taken into account in CFD, leading finally to a very good agreement.

---

**Fig. 3.** Representation of the air in blue (above the free surface), the water fluid regions in red, and the two simulated domains: either starting with the SEN inflow (region 1: a short computational domain), or including the tundish (region 2: both regions visualize the complete geometry).

**Fig. 4.** SEN outlet with PIV calibration target during caster filling. At the end, the calibration target is fully immersed.
Modeling

A free surface separating water and air appeared around the outlet of the SEN. In order to simulate accurately the position and the movement of this free surface, a computationally-intensive VOF (volume of fluid) model for Eulerian multiphase simulations was employed. All numerical details concerning the VOF implementation could be found in Hirt et al. [9], Brackbill et al. [10] and the STAR CCM+ Documentation [11]. This approach allowed a direct computation of the experimental setup. The volume flow out of the SEN depended fully on the height of the stopper. The numerically-predicted position of the free surface around the SEN was tracked and used as an indicator that physically realistic conditions were obtained by CFD. The CFD tool used to compute the short domain was ANSYS CFX-13.0. As the required computational power got much higher with the complete geometry, it was later necessary to switch to another industrial software (STAR CCM+ v11) due to license limitations. Both software used the same fundamental equations and models and rely on a finite-volume discretization. All settings and boundary conditions of the CFD are listed in Table 2 for the complete geometry involving Regions 1 and 2.

Due to the fundamental importance of this issue, two turbulence models were compared, the k-ω-SST model [12] and the Realizable k-ε turbulence model [13]. However, the results achieved with the latter model did not exhibit satisfactory agreement with the experiments, as also mentioned by Javurek et al. [14]. As a consequence, the turbulence model used as a reference in all further computations referred to the k-ω-SST model, since it showed the best agreement with the PIV measurements.

The wall boundary layer was resolved with a hybrid treatment called “all-$$y'$$ wall treatment” that emulated the low-$$y'$$ wall treatment (viscous sublayer $$y'< 5$$) for sufficiently fine meshes, and the high-$$y'$$ wall treatment (log-law layer prescribed for $$y' > 30$$) for coarse mesh. This method blended the turbulence quantities such as dissipation, production, stress tensor, etc. calculated by the other two approaches with an exponential weighing function [11]. The maximum $$y'$$ value in all simulations with the complete geometry referred to $$y' = 14$$. It was suitable for all-$$y'$$ wall treatment.

Data Post-Processing

In order to compare directly PIV measurements and CFD data, a simulation of the flow within 10 s of physical time was initially carried out. It provided formation of a fully-developed flow within the numerical model. The velocity field obtained by CFD was averaged for a further time period of 10 s with a frequency of 15 Hz, exactly as done for PIV data as slow oscillations were observed both experimentally and numerically.

The flow component orthogonal to the “laser” sheet was not taken into account in the comparison, since it could not be measured by PIV, even if it was obviously computed by CFD (Fig. 6).

Flow at SEN Outlet from Short Computational Domain

As already explained, the main region of interest was the flow entering the caster at the outlet of the SEN. Due to the considerable size of the complete water model, the first CFD simulations considered only the lower part of
the model numbered as region 1. This short computational domain started within the SEN, and did include neither the tundish, nor the stopper. This led to enormous savings in terms of computing time and memory. Obviously, the effect of the flow exiting the tundish around the stopper head could not be represented with this short domain. Instead, constant inlet conditions were assumed at the entry of the vertical part of the SEN.

Considering that the length $L$ of the SEN after the $12^\circ$ bend was 1.5 m, with a constant inner diameter $D$ of 56 mm, a ratio $L/D \approx 27$ was obtained prior entering the caster. This should be sufficient [5] to obtain fully-developed turbulent flow conditions corresponding to $L/D > 25$. According to alternative recommendations [6], this condition could be calculated from the Reynolds number $Re$ as shown by Eq. (1):

$$L > 4.4 \left(Re\right)^{1/6} D$$

Using the calculated value of $Re$ in Table 2, a minimum length of 1.68 m would be required. It was close to the available 1.5 m. If the flow would be really fully developed at the outlet of the SEN, the shorter computa-

---

**Table 2. CFD settings, flow properties and models.**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Trimmed Mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cell number</td>
<td>~ 5 000 000</td>
</tr>
<tr>
<td>Mesh adaption</td>
<td>Yes</td>
</tr>
</tbody>
</table>

**Boundaries**

| Inlet          | Prescribed mass flow rate |
| Outlet         | Prescribed mass flow rate |
| Mass flow rate | 3.5 kg s$^{-1}$           |
| Density        | 1 000 kg m$^{-3}$         |
| Walls          | no slip                   |

**Physical models**

| Reynolds-Averaged-Navier Stokes | Implicit Unsteady |
| Adaptive time step              | Target CFL number=15 |
| Eulerian Multiphase             | Volume of Fluid (VOF) |
| Surface tension                 | 0.074 N m$^{-1}$     |

**Turbulence at SEN**

| Mean velocity within SEN         | 1.43 m s$^{-1}$ |
| Inner diameter of SEN            | 56 mm           |
| Reynolds number within SEN       | 99 947          |

**Turbulence Models (two tested)**

| Realizable k-$\varepsilon$   | All y+ wall treatment |
| k-$\omega$-SST               | All y+ wall treatment |

---

**Fig. 6. Exemplary CFD result showing the average velocity magnitude (colors) together with the streamwise velocity component (vectors). It highlights the strong jet observed at SEN outlet.**
A vertical line (shown as white line in Fig. 9) 2 cm after the SEN outlet was selected in order to check the agreement between the measurements and the numerical predictions in a quantitative comparison. The corresponding comparisons were plotted together with further results later in Fig. 12. The comparison between PIV measurements (the line with square symbols) and the first CFD results in the short computational domain (dotted and dash-dotted blue lines) in Fig. 12 reveals a very poor agreement; the shape of the profiles differ, with a very asymmetric result for CFD compared to nearly symmetric measurement data. Quantitatively, the peak velocity was much lower in the simulation. It was initially assumed that these large differences resulted from the assumed plug-flow velocity profile used as inflow boundary condition in the short domain (region 1).

Attempts were made in the subsequent simulations to take the effect of the real flow entering the SEN into account by considering a swirling flow as an inlet boundary condition. For this purpose, 10% radial and azimuthal components in relation to the axial flow velocity were prescribed at SEN inlet.

Using these inflow conditions, a velocity profile of a completely different shape was obtained by CFD, as shown by the bold dashed blue line in Fig. 12. It revealed indeed the sensitivity of CFD to the inlet boundary condition. However, the overall agreement was still unacceptably poor. Thus, simulating the whole water model appeared to be necessary for improving the prediction.

Simulation of the Complete Geometry

After simulating the water model in its entirety, including the tundish and the stopper, the poor agreement observed previously could be explained. Very complex flow structures and vortices were found around the stopper. They were still visible within the SEN as shown in Fig. 7. The complexity of the flow around the stopper could not be simply represented by a standard inflow condition as observed by Greis et al. [15] and He et al. [16]. As a consequence, the entire water model, as marked by regions 1 and 2 in Fig. 3, had to be systematically considered, as done in the rest of this work. It encompassed the ladle shroud, the tundish, the SEN with its bend, and the inner part of the belt caster (the mold). The height of the stopper was kept constant at 3.65 mm in all experiments.

RESULTS

Considering the whole geometry, the features of the flow can now be investigated in details. In particular, CFD reveals that the effect of the stopper and of the particular geometry around the stopper tip (the constriction area) extends far downstream. A noticeable acceleration (with corresponding decrease in pressure – not shown) occurs around the stopper, as seen in Fig. 7. The corresponding high-speed region generated downstream of the stopper still visibly impacts the flow after the bend of the SEN, see Fig. 8. Homogeneous flow conditions are not completely retrieved prior to SEN outlet.

The flow experiences acceleration due to the bend of...
the SEN as shown in Fig. 8. It returns to a fully-developed symmetrical pipe flow profile shortly prior to SEN outlet.

As in the experiment, CFD reveals slow pulsations in the flow associated with oscillations at the level of the water/air-interface around the SEN.

This fluctuation leads to a wavy 3-dimensional profile of the jet leaving the SEN (shown for instance in Fig. 9 when looking at the jet boundaries). This also explains a posteriori why the first CFD simulations using a steady-state solver never reached convergence.

As a consequence, the instantaneous fields differ noticeably from the results obtained by averaging over an interval of 10 s (Fig. 11). This highlights the observation time importance both in respect to the simulations and the experiments carried out.

In order to check the effect of the turbulence model, exactly the same simulation is repeated with the Realizable $k$-$\varepsilon$ turbulence model. In that case, a very strong and long-lasting swirling flow is observed in the SEN. It has a persisting swirl component at SEN outlet induced by the bend after the stopper. This results in a non-symmetrical velocity profile with a peak velocity on the lower side (Fig. 13), which is not observed at all in the experiments.

The comparison of Fig. 12 and Fig. 13, leads to the following observations:

- A satisfactory agreement between measurement...
The resulting agreement is good only in case of taking into account the time-averaged data. The instantaneous fields cannot be properly compared due to the flow pulsations at a low frequency but with relatively large amplitude.

- The k-ω-SST turbulence model delivers a much better prediction than the k-ε model realized in this case.
- The orthogonal velocity profile shows a back flow region close to every wall.

A similar comparison along the horizontal plane can
be seen in Correlations section and confirms the previous findings presented in Fig. 14, i.e. the comparison of the velocity magnitude obtained from CFD with the k-ε-SST turbulence model (color) and PIV (points) in the vertical comparison plane (Fig. 14, Fig. 15 and Fig. 16).
The entire cut planes are compared with each other aiming to quantify with a single number the differences between the obtained data sets. Thus, the data set from the CFD calculation (Fig. 16, left) is placed over the data set from PIV (Fig. 16, right) calculation at the same location providing a direct comparison illustrated in Fig. 14.

Then, the similarity between the two velocity fields can be quantified by a single value using a two-dimensional correlation [17, 18]. Such correlations are computed both in the vertical and in the horizontal measurement plane using either the $k$-$\omega$-SST turbulence model or the Realizable $k$-$\varepsilon$ model in CFD. As a consequence, four values are obtained. The correlation is illustrated in Fig. 15 for the vertical plane when using the $k$-$\omega$-SST turbulence model. An excellent correlation coefficient of 99.5% is obtained in this case.

**Correlations**

The entire cut planes are compared with each other aiming to quantify with a single number the differences between the obtained data sets. Thus, the data set from the CFD calculation (Fig. 16, left) is placed over the data set from PIV (Fig. 16, right) calculation at the same location providing a direct comparison illustrated in Fig. 14.

Then, the similarity between the two velocity fields can be quantified by a single value using a two-dimensional correlation [17, 18].

Such correlations are computed both in the vertical and in the horizontal measurement plane using either the $k$-$\omega$-SST turbulence model or the Realizable $k$-$\varepsilon$ model in CFD. As a consequence, four values are obtained. The correlation is illustrated in Fig. 15 for the vertical plane when using the $k$-$\omega$-SST turbulence model. An excellent correlation coefficient of 99.5% is obtained in this case.

---

**Fig. 14.** Comparison of the velocity magnitude obtained from CFD with $k$-$\omega$-SST turbulence model (color) and PIV (points) in the vertical comparison plane.

**Fig. 15.** A correlation between CFD ($k$-$\omega$-SST) and PIV for the vertical comparison plane. The perfect agreement corresponds to the red diagonal.

**Fig. 16.** Time-averaged velocity fields obtained by CFD with $k$-$\omega$-SST model (left) and PIV (right). The color scale and the dimensions are identical.
Overall, the comparison between CFD and PIV is very good for all four cases leading systematically to correlation coefficients quite close to 100 %. The only case in which a noticeably lower correlation is observed refers to the vertical comparison plane when using the realizable k-ε model. This verifies again the superiority of the k-ω-SST turbulence model. Note however, that almost no difference is observed in terms of correlation in respect to the comparison along the horizontal plane. This shows how important it is to involve several data sets in the analysis.

It is observed that the comparison points are slightly shifted towards the left of the diagonal throughout nearly the entire velocity range although the correlation shown in Fig. 15 results in an excellent correlation factor of 0.9954. The velocity of the PIV measurements is therefore almost always higher than that of the CFD simulation, which – when integrated – would result in a higher volume flow rate. Checking this point, it is indeed observed that the average volume flow rate in the water model is 12.7 m³/h, while that in the CFD calculation lay amounts to 12.5 m³/h. This minor relative difference (1.5 %) might be attributed to the very slight differences in the stopper opening between CFD and the experiment, as the stopper position determines the overall flow-rate. The position accuracy of the stopper is followed by microcontrollers and is thus very high. A mismatch of about 150 μm would be sufficient to explain the observed difference, as the tests have shown that a change in the stopper height by only 100 μm leads to an increase of the volume flow of 135 l/h.

CONCLUSIONS

This extensive study combining non-intrusive experimental measurements by Particle Image Velocimetry in a large-scale water model and CFD simulations using two different domain sizes and turbulence models reveals many interesting findings:

- As a consequence, it is necessary to take into account the stopper and its surroundings (the tundish) in the simulation; using a shorter computational domain leads to completely wrong flow features in the mold.
- Very large computational times are required in whole installation simulation because a large volume must be discretized with a sufficiently fine mesh, while describing properly the free-surface flow in the caster in a time-dependent frame.
- Though the overall configuration is steady (a constant stopper opening), both the experiment and the simulation results reveal flow oscillations at a low frequency but relatively large amplitude. They are probably connected to oscillations of the free surface at the entry of the caster. The comparison between the simulations and the experiments is therefore meaningful only for time-averaged data (averaged over at least 10 s).
- The comparison of CFD and the experiments shows that the k-ω-SST turbulence model delivers a better flow prediction compared to that of the realizable k-ε model. Therefore, the former model is recommended for further studies.
- Finally, taking all those recommendations into account, a good agreement is observed between the computational results delivered by CFD and those of the experimental measurements.

This closes successfully the validation step of the developed CFD approach. The latter is used in a further study to investigate the effect of the moving belts on the flow within the caster. It will provide the determination of the movement impact on the homogenization length.

REFERENCES

4. Twin-belt continuous caster with containment and cooling of the exiting cast product for enabling


