Flow Characteristics in a Blast Furnace Trough

Qinglin HE, Geoff EVANS, Paul ZULLI1), Francis TANZIL2) and Bob LEE2)

Department of Chemical Engineering, University of Newcastle, Callaghan NSW 2308, Australia.
1) BHP Steel Research Laboratories, Port Kembla NSW 2505, Australia. 2) BHP Steel’s Flat Products Division, Port Kembla NSW 2505, Australia.

(Received on February 7, 2002; accepted in final form on April 16, 2002)

The aim of this study was to better understand flow characteristics in the taphole stream impingement region of a blast furnace trough, and its effect on localized trough refractory wear. A 1/5th scale perspex model was used, and oil and water were adopted to simulate the molten iron and slag, respectively. Velocities and turbulence intensities in the region adjacent to the trough wall were measured by means of laser doppler velocimetry (LDV). This study highlighted the entrainment of bubbles by the impinging taphole stream, resulting in a buoyancy-driven flow pattern within the trough. The identified buoyancy-driven flow resulted in high velocities and turbulence intensities in the region where maximum refractory wear occurred. Methods for minimizing the influence of the buoyancy-driven flow, and resultant high velocities and turbulence intensities are proposed in this paper.

KEY WORDS: ironmaking; blast furnace; trough refractory wear; oil/water modeling; LDV.

1. Introduction

Significant savings in trough refractory costs can be achieved through measures to control wear in regions of high erosion. Statistical analysis of plant data at BHP Steel’s Flat Products Division, Port Kembla, Australia, has shown that the wear rate of the blast furnace trough refractory in the impingement region of the taphole stream is approximately three times greater than that further down the trough. Consequently, the local wear in the impingement region determines the period between major expensive repairs and the life of the campaign. The same refractory wear pattern in the blast furnace trough has been also reported in literature.1–4) Though a number of chemical reactions take place in this metallurgical system (slag-metal-refractory-air), the non-uniform wear profile along the trough wall would suggest that these reactions are unlikely to be rate (wear) controlling steps. It is believed that the localized refractory wear is mass transfer controlled, and is strongly influenced by the fluid flow conditions in the impingement region.

Most studies5–9) in the literature have focused on chemical erosion of the trough lining from the molten slag and iron. Some work1–3) has been carried out to reduce the refractory wear by changing flow conditions in the trough. This was carried out by altering the trough geometry, such as the floor slope, width, and wall angle, and taphole stream conditions including the drop height, impact angle and exit velocity. However, no detailed studies have been conducted with respect to flow characteristics in the blast furnace trough, in particular, in the impingement region. Consequently, there is lack of understanding of the fluid flow behaviour in the trough and its effect on the localized refractory wear.

In a previous study,10) the trough refractory wear in the region above the liquid level and on the trough cover was found to be directly linked to the flow characteristics of the taphole stream. In this study, the fluid flow within the trough, and subsequent localized refractory wear in the region below the liquid level, was investigated. A 1/5th scale perspex model was used, with oil and water as test fluids simulating the molten iron and slag, respectively. The fluid flow behaviour was investigated by measuring velocities and turbulence intensities in the region adjacent to the wall using laser doppler velocimetry (LDV). The effect of a change in the trough lining profile due to refractory wear on the fluid flow in the trough was studied by comparing the experimental measurements from a standard (new) trough with that from a worn one. The influence of the taphole angle on fluid flow was also examined. A method to reduce the trough refractory wear was proposed.

2. Experimental

2.1. Apparatus

Figure 1 shows a schematic of the experimental apparatus used in this study. The perspex trough model was 1/5th scale of the plant trough. Water and hydraulic oil (Shell, Tellus Oil C10) were used to simulate molten metal and slag respectively. Water and oil were pumped through separate lines and mixed at the nozzle (taphole), generating a two-phase (water/oil) taphole stream entering the trough. The ratio of oil and water volumetric flowrate was 1 : 0.747 (based on a plant metal/slag weight ratio of 1 : 0.29, and incorporating the density difference (ρmetal = 6 700 kg/m³; ρslag = 2 600 kg/m³) between the two phases). Oil separated
from the denser phase while traveling along the trough due to the density difference, and flowed into the oil tank via the 'slag' runner. Water flowed through the gap underneath the separating baffle, and then into the water tank. The tap-hole diameter was 0.0104 m ID, and tap-hole angle was 10 degree for all the tests, except those examining the effect of tap-hole angle on the fluid flow in the trough. The width of the worn trough model was 0.265 m at its widest point, with a gradual reduction to the dimension of the standard trough over the impingement region. Experimental conditions are given in Table 1. The choice of the water flowrates in the table was based on the similitude analysis described in the Section 3.

2.2. Measurement

Experimental program involved two separate systems. The first system involved water/oil to simulate metal/slag within the blast furnace trough. In this setup, characteristics of the three-phase (metal, slag and entrained gas) flow in the impingement region of the taphole stream was qualitatively investigated. In particular, identification of the buoyancy-driven flow and mixing of the three phases was determined. Initially, it was planned to undertake LDV measurements with the water-oil system to quantify the flow pattern. However, due to difficulties in data acquisition, caused by the oil phase adhering to the trough wall, the LDV measurements were not undertaken in the three-phase system. The inability to take these measurements was not a major drawback, given that it was found that the water-only system, which was used for LDV measurements, as described below, behaved essentially the same as for the water-oil system (see Sec. 4.1).

The second system, involving water only, was used to undertake LDV measurements, in which, the slag phase was not considered due to the reason described above. The LDV measurements were carried out in a grid (6×10) pattern, beginning at 0.8 m from the taphole. The LDV measurement area is shown in Fig. 1, i.e. 0.45 m along the trough and 0.05 m down the wall. The top of the measurement window was 0.01 m below the free surface. Three-dimensional (3-D) LDV measurements were conducted in all the tests except those examining the effect of taphole angle in which two-dimensional (2-D) LDV measurements were carried out. Mean velocity and root mean square (RMS) velocity (i.e. velocity standard deviation, which is a measure of turbulence) were recorded for each grid point. The LDV measurements were performed in 3 planes parallel to the trough wall. The planes were at 0.002, 0.01 and 0.02 m away from the wall, respectively.

3. Similitude

The laboratory model was constructed based 1/5th geometric similitude with the full-scale plant trough. The main dimensions are shown in Fig. 1. The volumetric flowrate of metal (water) and slag (oil) through the laboratory model was based on a Froude number similarity criteria for the taphole stream, where the Froude number (Fr) is defined as:

\[
Fr = \frac{u^2}{gl} \tag{1}
\]

where, \(u\) is the taphole stream (jet) velocity and \(l\) is the characteristic dimension (diameter) of the taphole.

These were two reasons for basing the laboratory model throughput on a taphole stream Fr number similarity criteria. Firstly, it provided similar jet characteristics, in terms of surface roughness and trajectory into the trough, to that observed in the plant. Secondly, the jet Fr number has been shown to be a good correlation for relating the quantity of gas, \(Q_G\), entrained by a plunging liquid jet,\(^{11}\) i.e.

\[
\frac{Q_G}{Q_J} = 0.04Fr_{40}^{0.26} \left( \frac{L_J}{D_J} \right)^{0.4} \tag{2}
\]

where, \(Q_J\) is the volumetric flowrate of the jet, and \(L_J\) and \(D_J\) are the free length and diameter of the jet, respectively.

Table 1. Experimental conditions.

<table>
<thead>
<tr>
<th>Variables</th>
<th>LDV Measurement (water/oil system)</th>
<th>Flow Observation (oil/water/air system)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water flowrate (×10⁴ m³/s)</td>
<td>2.24</td>
<td>1.79 – 2.69</td>
</tr>
<tr>
<td>Oil flowrate (×10⁴ m³/s)</td>
<td>0</td>
<td>1.31 – 1.96</td>
</tr>
<tr>
<td>Oil layer thickness (mm)</td>
<td>0</td>
<td>35 – 70</td>
</tr>
</tbody>
</table>

Fig. 1. A schematic of the experimental apparatus (not to scale).
It is important for the laboratory model to have similar gas entrainment characteristics to that in the plant, because it is found (see Sec. 4) that the entrained bubbles, and the subsequent buoyancy-driven flow in the taphole stream impingement region, strongly influences the liquid motion in the trough.

The physical properties of the liquids used in both the plant systems and the laboratory model are shown in Table 2. It can be seen from Table 2 that the mixture (metal and slag) kinematic viscosity is very similar for both the laboratory model and plant systems. On this basis the fluid motion characteristics, such as energy dissipations and boundary layer development at the wall of the trough, should be similar. It is also worth noting that the kinematic viscosity of water is also of the order of $10^{-6}$ m$^2$/s, which is not too dissimilar to the water/oil system. The implication of this is that it is a reasonable approximation to use water only in the laboratory model, which make 3-D LDV measurements possible.

4. Results and Discussion

4.1. Flow Pattern in the Impingement Region

It was observed from the laboratory study that a considerable amount of air was entrained by the taphole stream from its surrounding medium into the liquid bath, as illustrated in Fig. 2. The entrained bubbles rose to the surface approximately 0.2 m from the taphole stream landing point, resulting in an upward flow in the bubble plume region. The location at which the bubbles escape is referred to as ‘bubble resurfacing point’, and the landing point of the taphole stream on the liquid surface is called impingement point in this paper. The taphole stream impingement region is defined, as marked in Fig. 2. The up-moving liquid changed its direction and turned into horizontal flow near the surface. The surface flow moving in the transverse direction impacted on the trough wall and then moved downward along the wall, as shown in Fig. 2. In fact, the rising bubbles, rather than the impinging stream, dominated the fluid flow in the impingement region. The flow pattern in the impingement region is essentially a gas buoyancy-driven flow caused by the rising bubbles. The rising bubbles entrained the surrounding liquid into the bubble plume. As a result, a slow reverse flow was generated near the bottom of the trough in the impingement region.

The surface flow pushed the oil phase away from the impingement region. As a result, a thick oil layer (thicker than average) formed in the ‘slag’ runner end of the trough, a very thin layer of oil remained near the furnace end, and no oil layer left in the impingement region, as seen in Fig. 2. It was virtually a ‘single’ phase flow in the impingement region, though a mixture of the three phases (water/oil/air) existed in the region. This means that flow behavior in the impingement region in the two-phase (water–oil) modeling is expected to be the same as, at least very similar to, those observed in the water-only system.

It is worth mentioning here that the same fluid flow phenomena as seen in the modeling experiments were also observed in the industrial trough in the plant trials of the previous study. The entrained bubbles resurfaced at a distance downstream the taphole stream impingement point. The distance was less than what would be scaled up from the modeling result by using the geometric scale factor. This was because of the larger liquid-gas density difference in the plant case than in the model. The slag layer was observed to be pushed away from the impingement region, leaving exposed hot metal free surface. However, the extent of the hot metal exposure depends on the amount of the slag phase in the trough. The thicker the slag layer is, the smaller the exposure area.

Oil, as part of the water/oil mixture taphole stream, pene-
treated into the liquid pool, and then broke up into oil droplets, as shown in Fig. 2. The oil droplets penetrated more deeply into the bath than the entrained bubbles. They dispersed in the liquid pool more evenly across the trough width in the impingement region, while the entrained bubbles tended to concentrate in the central region (along the trough axial plane).

Figure 3 demonstrates typical velocity profiles near the trough wall in the impingement region for a standard trough, measured using LDV in the water-only system. The velocity profiles were measured in the planes 0.002, 0.01 and 0.02 m away from the wall respectively. The length of the velocity vectors is scaled to the magnitude of the velocity. The impingement point of the taphole stream and the bubble resurfacing point are also marked in the figure. It can be seen from the figure that the velocity vectors in the region in line with the bubble resurfacing point have a large angle relative to the trough wall, which confirms the flow observation that the liquid flow impacts on the trough wall in the region. There was large variation in the fluid velocities within the measurement window. Velocities in the region downstream of the bubble resurfacing point are an order of magnitude higher than those in the region close to the furnace end of the trough. The liquid near the wall was moving forwards along the trough (see Fig. 3(a)), while the liquid in the central and lower part of the trough was flowing backwards and towards the plume region (see Fig. 3(c)). The surrounding liquid was entrained into the plume region by the rising bubbles, resulting in the liquid in the surrounding region flowing towards it.

The velocity and turbulence energy contours are shown in Figs. 4 and 5, respectively, for the plane 0.002 m from the wall. The turbulence energy is defined as $0.5(u_{rms}^2 + v_{rms}^2 + w_{rms}^2)$; where $u_{rms}$, $v_{rms}$ and $w_{rms}$ are the root mean square velocity in three coordinate directions, respectively. As seen in the figures, the maximum turbulence energy took place in the region in line with the bubble resurfacing point, while the maximum velocity occurred in the region further down the trough. Both the maximum values took place at some distance away from the impingement point of the taphole stream. This strongly suggests that the
The fluid flow in the impingement region is dominated by the rising bubbles, rather than the taphole stream itself.

The maximum velocity and turbulence energy for the measurement region are shown in Figs. 6 and 7, respectively, as a function of distance from the wall. The results are also noted in Table 3. It can be seen from the figures that when the measurement plane was moved away from the wall, the maximum velocity decreased, while the maximum turbulence energy showed the opposite trend. The velocity variation with distance from the wall can be explained by the flow field (outside the boundary layer) generated by an impinging jet on a solid surface. The increase in turbulence with distance from the wall indicates that the turbulence within the trough was dominated by the rising bubbles. The farther it is from the wall, i.e., closer to the bubble plume region, the higher the turbulence is.

### 4.2. Effect of Taphole Angle

The effect of taphole angle on the fluid flow in the trough was studied. 2-D LDV measurements, i.e., the two velocity components parallel to the wall, were carried out at different taphole angles, 8°, 10°, and 12°. The maximum velocity and turbulence energy for the measurement window at 2 mm from the wall of the standard trough are compared in Table 4. It can be seen that the taphole angle did not significantly affect the maximum velocity (5% variation) and turbulence energy (8% variation). However, the taphole angle did affect the location of the impingement point of the taphole stream and hence the bubble resurfacing point. The taphole stream landed in the trough at 0.82, 0.87, and 0.91 m from the taphole exit for taphole angles 8°, 10°, and 12°, respectively. The velocity profile was just simply shifted away from the taphole as the taphole angle increased.

### 4.3. Fluid Flow in a Worn Trough

Over time, the trough refractory lining wears to form a curvilinear profile. In order to understand how the change affects the fluid flow in the trough, LDV measurements were carried out in a trough model with a belly-shaped impingement region. This simulates a worn trough refractory profile. Figure 8 shows a velocity profile near the wall in the impingement region of a worn trough. The measurement plane had a curved shape, parallel to and 2 mm away from the worn wall. Comparing Figs. 8 with 3(a), it can be seen that the velocity profiles are similar in the standard and the belly-shaped trough, though their magnitudes are different.

The results for maximum velocity and turbulence energy for the measurement area obtained from the belly-shaped trough are presented in Table 3, in comparison to those

---

**Table 3.** LDV measurement results under different conditions.

<table>
<thead>
<tr>
<th>Trough Design</th>
<th>Distance from wall (mm)</th>
<th>Max. Velocity (m/s)</th>
<th>Max. Turbulence Energy (m²/s²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard trough</td>
<td>2</td>
<td>0.568</td>
<td>0.0242</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>0.508</td>
<td>0.0266</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>0.461</td>
<td>0.0408</td>
</tr>
<tr>
<td>Belly-shaped trough</td>
<td>2</td>
<td>0.491</td>
<td>0.0147</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>0.419</td>
<td>0.0249</td>
</tr>
<tr>
<td>Standard trough</td>
<td>2</td>
<td>0.357</td>
<td>0.0109</td>
</tr>
<tr>
<td>with baffles</td>
<td>10</td>
<td>0.396</td>
<td>0.088</td>
</tr>
<tr>
<td></td>
<td>20</td>
<td>0.607</td>
<td>0.0452</td>
</tr>
</tbody>
</table>

---

**Table 4.** Comparison of LDV measurements between different taphole angles.

<table>
<thead>
<tr>
<th>Taphole Angle (degree)</th>
<th>Max. Velocity (m/s)</th>
<th>Max. Turbulence Energy (m²/s²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>0.322</td>
<td>0.0116</td>
</tr>
<tr>
<td>10</td>
<td>0.327</td>
<td>0.0118</td>
</tr>
<tr>
<td>12</td>
<td>0.337</td>
<td>0.0125</td>
</tr>
</tbody>
</table>
from the standard trough. It can be seen that both the velocity and turbulence energy near the wall were significantly lower in the belly-shaped trough. The maximum velocity and turbulence energy for the plane 2 mm away from the wall decreased by 13.6% and 39.3% respectively, in comparison with those for the standard trough design. The worn trough was wider and hence had a larger volume in the impingement region, in comparison with the standard one. The same amount of input energy from the rising bubbles dissipated into a larger liquid volume, resulting in a lower velocity and turbulence energy of the liquid in the region.

4.4. Effect of Baffles

The impact of the buoyancy-driven flow on the trough wall has a significant effect on the localized refractory wear in the region. As a countermeasure, two baffles of 0.03 m wide and 0.45 m long were horizontally placed on the walls along the trough, just below the free surface, with the aim of reducing the velocity and turbulence of the liquid near the wall (see Fig. 1). 3-D LVD measurements were performed with this configuration. The velocity profiles in the planes 0.002, 0.01 and 0.02 m from the wall are illustrated in Fig. 9. Comparing Fig. 9 with Fig. 3, it can be seen that the placement of the baffles had a significant effect on the fluid flow in the impingement region. The velocities near the wall (0.002 m away from the wall) are much lower in the baffled trough than the standard one, in particular, in the impact region of the buoyant-driven flow (between 200 mm and 300 mm). However, the velocities in the region away from the wall (0.02 m) are higher with the baffles than without them.

The maximum velocity and turbulence energy for the measurement window are presented in Figs. 10 and 11, respectively, along with those from the standard trough without the baffles. The results are also noted in Table 3. It is interesting to note that the variation of the maximum velocity with the distance from the wall shows different trends with and without the baffles. In the former case, the maximum velocity increased, while in the latter it decreased as the distance from the wall increased. This arose from different flow patterns generated in the two troughs. With the baffles in place, the gas buoyancy-driven flow was forced to turn downwards before reaching the wall. As a result, the velocity near the wall was significantly reduced, in comparison
with the case without the baffles in which the liquid flow directly impacted on the wall. These results indicate that the high velocity region was moved away from the wall when the baffles were in place. With respect to turbulence energy, because it was dominated by the rising bubbles, the same trends were obtained with and without the baffles as distance from the wall increased, as seen in Fig. 11. However, the turbulence energy level near the wall was significantly reduced with the baffled trough. The maximum velocity and turbulence energy in the plane 2 mm from the wall were reduced with the baffled trough by 37% and 55%, respectively, in comparison with the one without the baffles.

5. Implication of Plant Implementation

As discussed earlier, a large number of bubbles are entrained into the liquid metal pool by the taphole stream, and they result in a gas buoyancy-driven flow pattern in the impingement region of the taphole stream in the blast furnace trough. In other words, the rising bubbles dominate the fluid flow behaviour in the region. As a result, high turbulence and large velocities occur in the region downstream, and near the bubble resurfacing point. In an operating trough, the maximum refractory wear rate is expected to occur in the region downstream and near the bubble resurfacing point, rather than in the region close to the taphole stream impingement point. This is clearly demonstrated in Table 5. As seen from the table, the taphole stream landed in the trough at around 3.5 m from taphole, while the maximum wear rate of the trough refractory lining occurred at some 5 m from taphole.

It has been found from the LDV measurements that the velocity and turbulence energy near the wall are lower in a worn trough than in a new one under the same conditions. Hence, the refractory wear rate in the plant trough is highest at the beginning of the trough life campaign, and gradually reduces over the life of the trough.

It has been demonstrated that the use of the baffles is a very effective way to reduce the velocity and turbulence energy near the trough wall. The high velocity region is shifted away from the wall when the baffles are in place. As a result, a reduction in the trough refractory wear rate would be expected in the plant trough with the baffles installed. Plant trials using scaled up baffles are being carried out at BHP Steel’s Flat Products Division.

Changing taphole angle within the experimental conditions of this study is not expected to lead to any change in trough refractory wear rate. However, the position of the localized refractory wear will change, moving away from the taphole as the taphole angle is increased.

6. Conclusions

From this study, it may be concluded that:

- The flow behavior in the impingement region of the taphole stream in the water–oil modeling is expected to be the same as, or at least very similar to those observed in the water-only system. The oil in the impingement region is pushed away by the buoyancy-driven flow resulting from the rising bubbles entrained by the taphole stream. It is virtually a ‘single phase’ flow, though there exist a mixture of water/oil/air in the region.
- A gas buoyancy-driven flow has been identified. As result of its direct impact on the wall and subsequent high liquid velocity near the wall, the gas buoyancy-driven flow is believed to have a significant effect on the localised wear of the trough refractory. Placing baffles on the walls just below the free surface is an effective countermeasure for the gas buoyancy-driven flow. It can significantly reduce both velocity and turbulence energy near the walls. Compared with the standard trough, the reduction in the maximum velocity and turbulence energy was 37% and 55%, respectively. This would result in a decrease in the trough refractory wear in the region.
- Velocity and turbulence energy near the walls are lower in a worn trough than in a new one. This means that the wear rate of trough refractory is highest at the beginning of a trough life campaign, and gradually decreases as the time of trough in service increases.
- There are no significant effects of the taphole angle on the fluid flow in the trough, except the difference in location of the impingement point. The flow pattern simply shifts down the trough as the taphole angle increased within the taphole angles considered in this study.

Acknowledgements

We would like to express our appreciation to Dr. Ron Roberts for his assistance in the LDV measurements. Thank also goes to Mr. Paul Markham for his help in plant data collection.

REFERENCES


| Table 5. Comparison of trough refractory wear rate with LDV measured results. |
|----------------------------------|-----------------|
| Distance from taphole, m         | 3 | 3.5 | 4 | 4.5 | 5 | 5.5 | 6 | 7 | 9 |
| Trough Refractory wear, mm/HM$^a$| 2.24 | 3.51 | 3.82 | 3.60 | 2.93 | 2.12 |
| LDV measured velocity, m/s       | 0.17 | 0.25 | 0.33 | 0.52 | 0.47 |
| LDV measured turbulence energy, m/s$^a$ | 0.059 | 0.014 | 0.020 | 0.013 | 0.075 |
| Impingement point of taphole stream$^{10}$ | X |
| Maximum refractory wear point    | X |

a: HM – tonne of hot metal;
b: locations at which LDV measurements were carried out have been scaled up by the geometric scale factor of 5.