MODELLING STUDY OF JET- METAL INTERACTION IN LD PROCESS

M. K. Mondal[1], Logachander.K[2]

[1]*Assistant Professor, National Institute of Technology Durgapur, Durgapur, West Bengal, India*
[2]*Assistant Manager, BOF, SAIL-Durgapur Steel Plant, Durgapur, West Bengal, India*

Abstract

Water model experiments have been carried out in a 1/30th scaled down model of the 100 ton LD converter in order to investigate the effect of changing the lance height and the gas flow rate on the penetration depth of liquid with different exit diameters. It is found the penetration depth increases with decreasing nozzle diameter, decreasing the lance height and with increase the gas flow rate.

Gas jets impinging onto a gas–liquid interface of a liquid pool are also studied using computational fluid dynamics modeling, which aims to obtain a better understanding of the behavior of the gas jets. The gas and liquid flows are modeled using the volume of fluid technique. The governing equations in the axisymmetric cylindrical coordinates are solved by the CFD simulation using FLUENT. The computed results are compared with experimental result and it isfound a good match with all the data.

Keywords: *LD process, Water Modeling, Penetration Depth, Volume of Fluid, CFD.*

---***---

1. INTRODUCTION

The LD process uses supersonic oxygen jets that impinge on the metal bath surface and the impingement process promotes the refining reactions and slag formation. The LD has an extremely high refining rate partly because of the large amount of interfacial area created among the metal and slag. When blown oxygen strikes the steel melt a cavity is formed on the liquid surface. The chemical reaction takes place both in the cavity and in the foam or slag that is produced in the process. The slag contains bubbles and provides a large surface area for the oxidation. The depth and diameter of the cavity, heat and mass transport at the interface and in the liquid are important parameters in the process. Liquid circulation driven by the jet and cavity dimensions depend on the following process variables such as gas flow rate,distance from the nozzle to the liquid free surface,angle formed between the lance and the liquid free surface,shape, size and number of nozzles in the lance,density and viscosity of gas and liquid[1].

It is difficult to measure the above parameters in actual process conditions prevailing in BOF.Physical modeling, using water and transparent vessels, is an important tool to perform these studies in a laboratory scale at low cost and safe conditions. Also, Mathematical modeling is a tool able to simulate these complex systems.

Behera et al [2] studied the effect of the no nozzle (single hole, two holes and three holes) in jet penetration by cold model experiment. Patjoshiet al [3] concluded that depth of the depression and splashing are increased with increase in jet momentum and nozzle angles and decrease with lance height by the water model experiment. Banks et al [4] investigated the pressure distribution, cavity profiles and associated velocity profiles for round and planar jets. Nordqist et al [5] concluded that penetration depth is function of gas flow rate, nozzle diameter and lance height by the cold model study. Sharma et al. [6] concluded that the size of the cavity is controlled by the jet strength, while Koria and Lange [7] gave more importance to the lance free surface distance and gas pressure (impinging the liquid surface). Qian et al. [8] obtained measurements of depressions in water model having a lighter second liquid phase (corn oil or kerosene) on the top of water to simulate the presence of slag. Subagyo et al. [9] have correlated splashing phenomenon with the normalized Weber number. Peaslee and Robertson [10] developed a model using CFD commercial software FLUENT that describes liquid velocity fields in the vicinity of the impingement point. They also presented results on the splashing and oscillation of the free surface as a function of the lance angle. Gu and Irons [11] applied the volume of fluid (VOF) algorithm to describe a liquid free surface and to determine the velocity fields in a 3D fragment of a vessel driven by the action of a gas jet. In this workthere are no precise measurements of the cavity geometry and therefore only a qualitative comparison wasmade between model predictions and experimental photographs and besides they do not validate the predicted velocityfields and only showed a qualitative analysis of the flow patterns. Regarding the cavity geometry, Nakazono et al. [12] modelled the depth of cavity due the impingement of an oxygen jet in a free liquid iron surface using a force balance under vacuum and proposed a correlation that describes the jet penetration in the free surface. So behavior of jet is important for better understanding the process inside the LD (BOF) process.

In the present study 1/30th geometric scale model of a 100 ton capacity LD convertor has been used. The model was constructed of Perspex sheet. Simulation of the steel bath was carried out by using water and oxygen gas by air.For the model study of this type, where bath flow behavior is simulated, the modified Froude number is the basic criterion. This dimensionless number in the model should be same as that in actual converter. Modified Froude number is defined by equation (1).

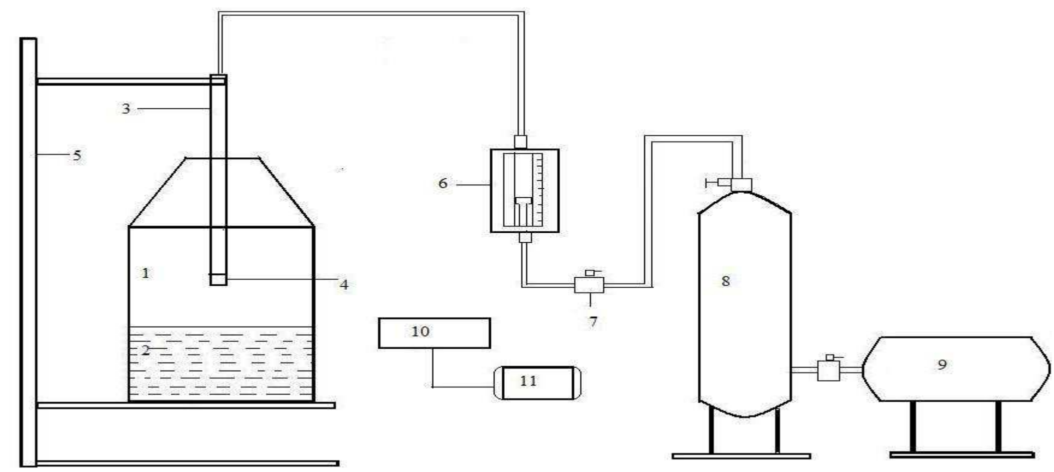
$$N_{Fr} = \frac{\rho_g V_0^2}{g(\rho_l - \rho_g)d_0} \quad \text{……………………. (1)}$$

Where, ρ_g is the density of the gas jet, V_0 is average gas velocity at nozzle tip, ρ_l is the density of the liquid and d_0 is the hydraulic diameter of the nozzle.

The objectives of this paper is to study the effects gas flow rate, nozzle diameter and lance height on depth of penetration and simulate the water model LD converter using k-ε turbulent and VOF model and compare this result with experimental.

2. EXPERIMENTAL SET UP AND MEASUREMENT

Experimental Set up is shown in Fig.1.The vessel was filled with clean water up to a height of 0.1m; front portion of the converter was fitter with transparent plastic sheet graph paper of 1 cm division in both the direction. A lance was lowered to a pre-fixed distance from the water surface and air jet at fixed flow rate was allowed to impinge on the surface. The crater formed on the bath surface due to the impact of air jet, at chosen lance height and fixed inlet flow rate video photographed was taken. The lance height was varied from 0.1 m, 0.11 m and 0.12 m, nozzle diameter was varied 1.5 mm, 2mm 2.5mm and 3mm and the flow rates were varied from 30 Lt/min to 70 Lt/min.



1. Converter vessel 2. Water bath 3. Lance 4. Lance Tip(nozzle) 5. Supporting Stand 6. Flow meter 7. Air control valve 8. Accumulator tank 9. Air compressor

Fig.1: Experimental set up for water model Experiment

The deformation created at the liquid surface was captured by video photography. Finally the digitized versions of the photographs were used to measure the diameter and height of the crater by computer.

3. MATHEMATICAL MODELING

The computational domain is shown in Fig.2 and the following assumptions have been adopted.

1) All fluids areNewtonianand incompressible fluid and constant molecular viscosity.
2) Axis-symmetry and unsteady
3) No masssources are taken into account
4) Isothermal flow and free surfaces in the water and air are assumed to be flat.

The jetting system is governed by the gas and liquid flows and can be modelled using the volume of fluid (VOF) technique, which is a fixed grid technique designed for two or more immiscible fluids where the position of the interface is of interest.In the VOF model the volume fraction of each of the fluids in each computational cell is tracked throughout the computational domain.

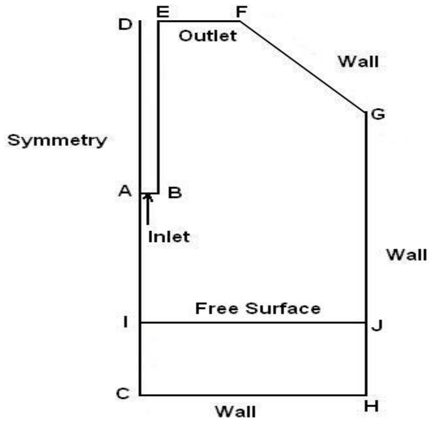


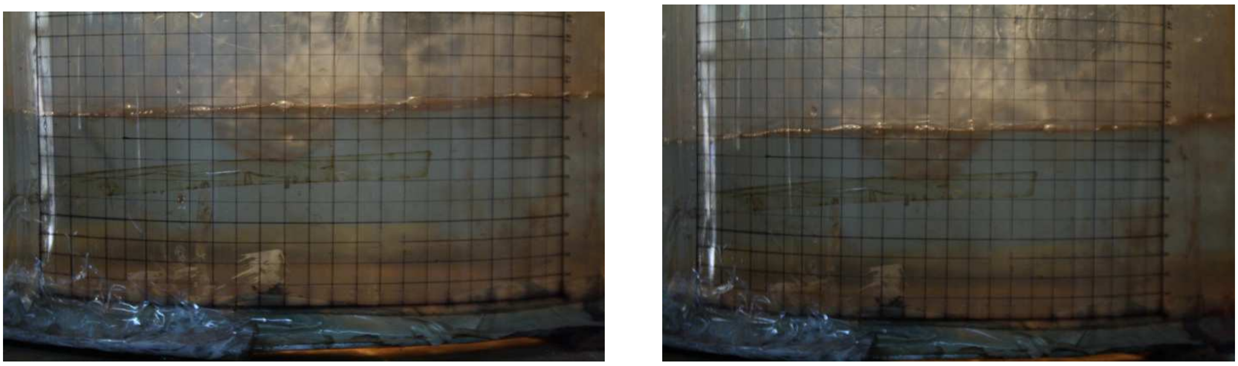
Fig.2: Schematic representation of the system for the purpose of mathematical modeling

In addition to the velocity and pressure, the volume fraction is also a variable of the flow field in the VOF technique and hence is contained in both the mass and momentum equations. Five types of boundary conditions were used to describe the flow field within the computational domain:

1. Top of the BOF which is opened to atmosphere is defined as pressure out let (boundary EF in Fig.2) boundary condition. This value was set to zero, i.e. assumed to be at atmospheric pressure.
2. Velocity inlet is defined at the nozzle hole (boundary AB in Fig. 2) and velocity is calculated from the gas flow rate. Volume fraction of air is one at the inlet.
3. Standard wall functions are employed to represent the near wall regions (boundary BE, FG, GJ, JH and CH in Fig. 2) and at the solid wall no slip condition is specified.
4. At the free surface (boundary IJ in Fig. 2), the normal gradients of the parallel velocity components, turbulent kinetic energy, kinetic dissipation rate and tracer concentration are set to be zero.
5. The axis boundary was used along centerlines DA, AI and IC. The radial velocity component V and the gradients of the other dependent variables were equal to zero.

For this model quad-map 0.011 mm strong grid luster was used along the central line of BOF converter and grid size of 1 mm was used away from the central line of BOF converter. Non uniform mesh/grid was created using Gambit (version 2.0.4). The governing partial differential equations were converted to algebraic equation by adopting staggered grid control volume technique. The governing equations were discretized using the second-order upwind scheme to achieve the bestaccuracy. The interpolation of the pressure values at the cell faces, using the momentum equation, was carried out using the PRESTO (Pressure Staggering Option) scheme, which improved the convergence rate and the stability of the computation. Pressure–velocity coupling was achieved by using the PISO (Pressure-Implicit with Splitting of Operators) algorithm with neighbor correction, which is highly recommended for both steady state and transient calculations on meshes with a degree of distortion. The governing equations were solved sequentially using the commercial Fluent® (versions 6.3.16) CFD software. Stable transient solutions were obtained with relatively small time steps, typically between 1 × 10-6and 1 × 10-4 s.

4. RESULTS AND DISCUSSION



(a) (b)

Fig.3: An example of cavity formation at 0.1m lance height and different gas flow rate of (a) 40 Lt/min and (b) 45 Lt/min

The series of photographs (such as Fig.3(a) and 3(b))has been taken for the investigation the effect of lance heights, gas flow rate and nozzle diameter. It is evident that the profile of the crater is the shape of a parabola. From those photograph depth of penetration are measured and the penetration depth with gas flow rate, lance height and nozzle diameter are plotted in Fig.4, Fig. 5 and Fig.6 respectively.

4.1 Influence of Gas Flow Rate:

As is clear from Fig.4 for a given nozzle diameter (0.00225 m) and a fixed lance height (0.1 m), the depth of penetration increase from 0.023 m to 0.033 m with increase in gas flow rate from 40 Lt/min to 70 Lt/min[5]. This result is in good accordance with the well established theory that an increase in gas flow rate causes the jet momentum to increase which displaces more volume of liquid, so higher is the depth of penetration

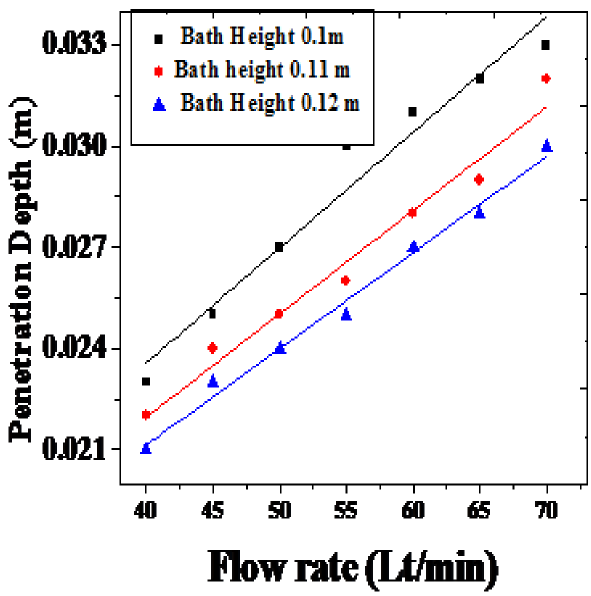


Fig.4: Depth of the penetration versus the gas flow rate at different bath height for 2.25 mm single hole nozzle

4.2 Influence of Lance Height:

The effect of changing the lance height for a given nozzle diameter and at a fixed gas flow rate is well indicated by Fig.5.As lance height is increased, the depth of penetration decrease due to the fact that jet after coming out of nozzle interacts with ambient atmospheric air and losses its force due to this interaction, as well as due to spreading, resulting on lesser is the penetration depth at greater bath height or lance height [5].

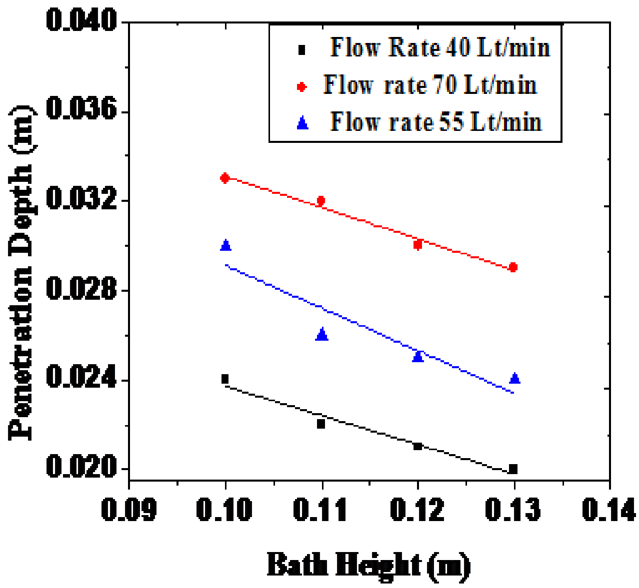


Fig.5: Depth of the penetration versus the lance height at different gas flow rate for 2.25 mm single hole nozzle

4.3 Influence of Nozzle Diameter:

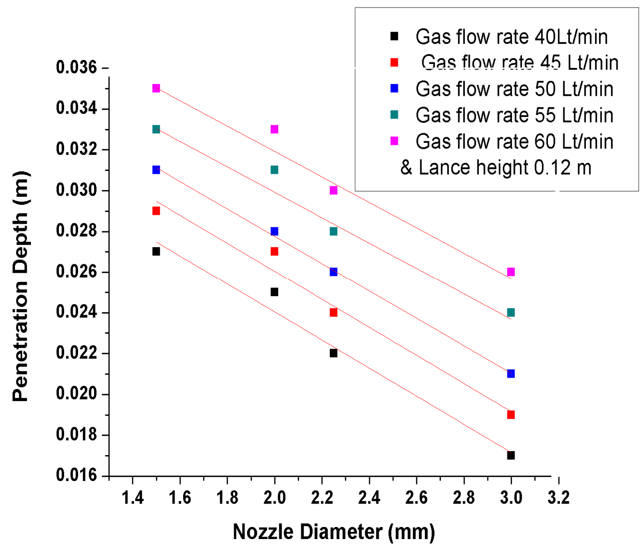


Fig.6: Depth of the penetration versus the nozzle diameter at different gas flow rate for 0.12 m lance height.

Fig.6 shows the influence of nozzle diameter on the depth of penetration. The smaller the nozzle diameter, more is the penetration depth at a given gas flow rate and fixed lance height. Smaller diameter nozzle tends to produce higher momentum for a given gas flow rate and hence give rise to deeper cavities [5].

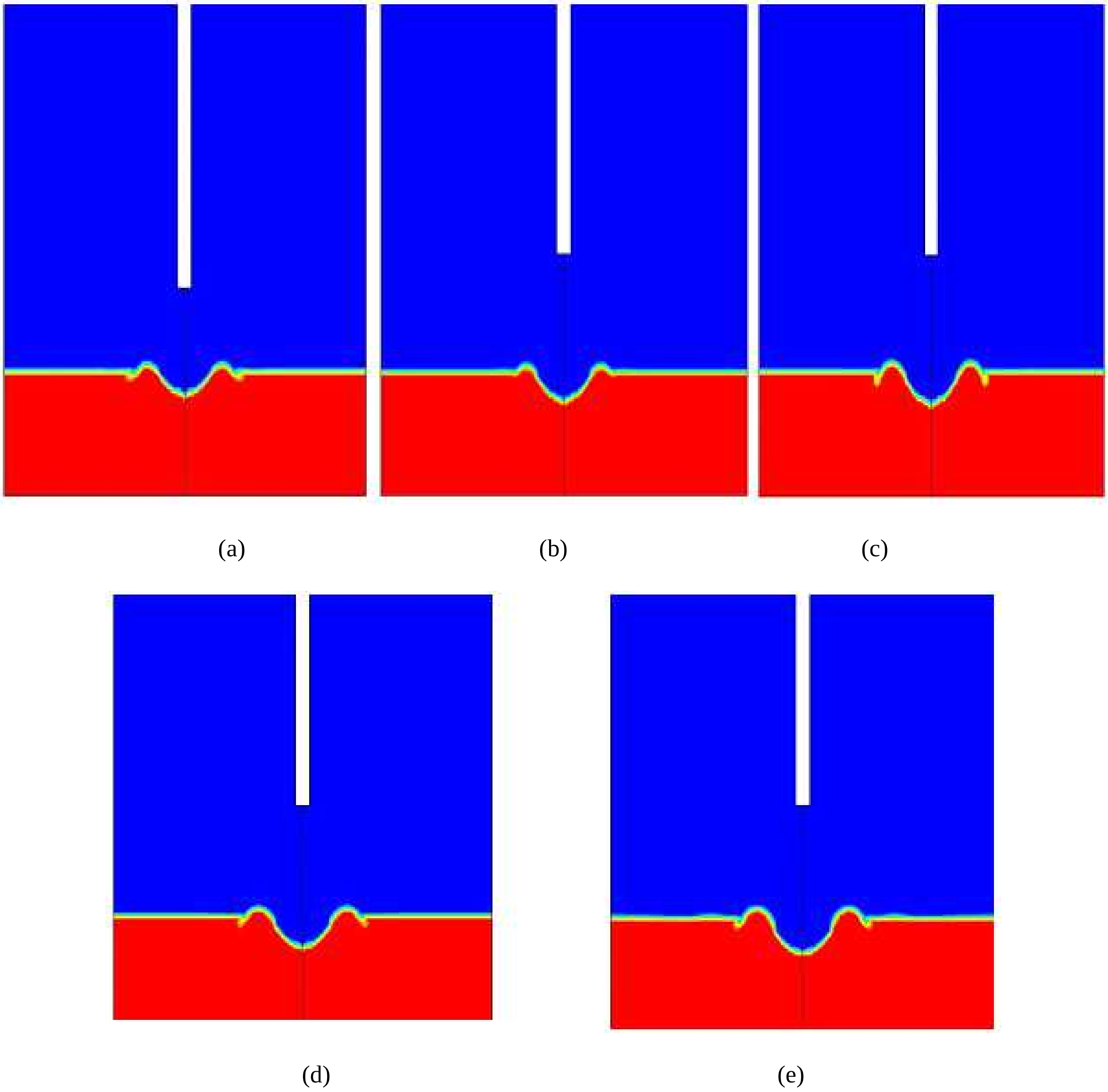


Fig.7: Gas-Liquid interface (free surface) profile at gas flow rate of (a) 40 Lt/min, (b) 45 Lt/min, (c) 50 Lt/ min, (d) 55 Lt/min and (e) 60 Lt/min when nozzle diameter of 2.25 mm and lance height 0.10 m.

Fig.7 shows the cavity profile predicted by the VOF model. The parabolic shape of the cavity has been formed due to impinging of air jet onto water pool using VOF model. From figure 7 depth of penetrations are measured and it is also observed that when gas flow rate is increase the penetration depth also increase at a fixed lance height and nozzle diameter as shown in the Fig.4.The computed penetration depths are compared with the experimental measured penetration depth and find a good match with all the results which is shown in Table 1.

Table 1: CFD and experimental penetration depth

	CFD	Experimental
Flow rate (Lt/min)	hc (cm)	hc(cm)
40	2.2	2.3
45	2.5	2.5
50	2.6	2.7
55	3	3
60	3.3	3.1

5. CONCLUSION

The major findings from this study are:

1. The penetration depth increases with increasing flow rate for a fixed nozzle diameter and a fixed lance height.
2. The penetration depth increases with the decrease in lance height for a fixed flow rate and a fixed nozzle diameter.
3. The penetration depth increases with the decrease in nozzle diameter for a fixed flow rate and a lance or bath height.
4. Computational fluid dynamics modeling is used to study gas jets impinging onto a gas–liquid interface of a liquid pool. The gas and liquid flows are modeled using the volume of fluid technique. The computed results are compared with experimental data and find a good match with all the data.

REFERENCES

[1] J. Szekely and N. J. Themekis, Jon Wiley & Sons, Inc. (1971) 716.
[2] R.C. Behera, A. Sarangi and S. Misra, Indian journal of Tech. 15 (1977) 419-422.
[3] A.K. Patjoshi, R.C. Behera, A. Sarangi and S. Misra , Iron & Steel int. February (1982) 27-32.
[4] B. Banks and D.V. Chandrasekhara, Journal of Fluid Mechanics, 15(1963), 13-33.
[5] A. Nordqist, N. Kumbhat, L. Jonsson, P. Jonssön, Steel Res. 77 (2006) 82–90.
[6] S.K. Sharma, J.W. Hlinka and D.W. Kern, Iron Steelmaker. 4(1977) 7–18.
[7] S.C. Koria, K.W. Lange, Steel Res. 58 (1987) 421–426.
[8] F. Qian, R. Muthasaran, B. Farouk, Metall. Mater.Trans. B 27B (1996) 911–920.
[9] Subagyo, G.A. Brooks, K.S. Coley, G.A. Irons, ISIJ International, 43 (2003) 983–989.
[10] K.D. Peaslee, D.G.C. Robertson, TMS, Warrendale, PA, 1994, pp. 1129–1145.
[11] L. Gu, G. Irons, 1999 Electric Furnace Conference Proceedings, Iron & Steel Soc., Pittsburgh, PA, 1999, pp. 269–278.
[12] D. Nakazono, K. Abe, M. Nishida, K. Kurita, ISIJ International, 44 (2004) 91–99.