An Assessment of Fluid Flow Modelling and Residence Time Distribution Phenomena in Steelmaking Tundish Systems

Anil KUMAR, Satish C. KORIA and Dipak MAZUMDAR

Department of Materials & Metallurgical Engineering, Indian Institute of technology, Kanpur, 208 016, India.
E-mail: kanil@iitk.ac.in

(Received on January 26, 2004; accepted in final form on May 10, 2004)

A summary of computational work reported in the literature on tundish hydrodynamics has been presented wherein, it is shown that a diverse range of both computational (e.g., nodal configurations, boundary conditions, inlet turbulence etc.) and physical parameters (e.g., size, number of strands, inlet mass flow rates etc.) were applied. Accordingly, the conclusions drawn were found to vary from one study to another. In the present work, an attempt has therefore been made to assess computationally the role of various mathematical model parameters. To this end, mathematical model results were validated against experimental measurements of Residence Time Distribution (RTD) parameters derived from water model tundish. Experimental measurements of RTD were carried out continuously by monitoring conductivity of water at the tundish exit port on a digital computer using a DAS interface. On the other hand, numerical calculations were carried out via the commercial CFD (Computational Fluid Dynamics) software FLUENT, 6.0. The combined experimental and computational study indicated that a sufficiently small grid resolution (control volume of the order of 10^-3 m^3) is necessary to arrive at a practical grid independent solution. Furthermore, a Reynolds stress model was found to simulate RTD in the system somewhat superior to the standard coefficient k-ε model. Through comparison of the predicted results with experimental measurements, a set of optimal mathematical model configurations was deduced. It was demonstrated that mathematical model configurations in this work is sufficiently reliable and robust as this leads to estimates of RTD parameters (e.g., t_{90}, t_{max}, t_{mean}) close to experimental measurements in a tundish with and without flow modifiers.

KEY WORDS: steelmaking; tundish; mathematical modelling; grid configurations; boundary conditions; turbulence modelling; computational assessment; experimental verifications.

1. Introduction

In recent years, the continuous casting tundish has evolved into a useful reactor for liquid steel refining. As such, it now has important roles to play over and above its traditional role as a buffer or steel distribution vessel. Thus a modern day steelmaking tundish is designed to provide maximum opportunity for carrying out various metallurgical operations such as inclusion removal, alloy trimming of steel, calcium doped inclusion modification, superheat control, thermal and particulate homogenization, collectively referred to as tundish metallurgy. Because of their associated techno-economic significance, tundish metallurgy operations have been investigated extensively in recent years. To this end, both physical and mathematical models have been applied. A good deal of these has already been summarised in a review presented recently by Mazumdar and Guthrie.

Since the chemical efficiency of tundish metallurgy operations is intricately related to the nature of fluid motion, consequently significant efforts have been made by researchers to investigate fluid flow phenomena in tundish systems. Apart from a few studies wherein, PIV or LDV was directly applied to uncover the flow experimentally, fluid flow phenomena in tundish has been generally studied via the pulse tracer addition technique and by estimating the various residence time distribution (RTD) parameters. It has been generally considered that ability of a computational procedure to simulate RTD phenomena realistically is a necessary indication of the adequacy of the underlying flow model. Accordingly, a great deal of studies on the numerical simulation of both flow and RTD characteristics on tundish systems has been reported in the literature. A summary of these is presented in Table 1. There, as seen, a diverse range of mathematical model parameters (e.g., grid size, boundary conditions, inlet turbulence parameters) were applied to compute flow phenomena in a wide range of tundish geometry. Essentially, the mathematical model configurations varied from one study to another (for example, compare the grid size distributions or the specified turbulence parameters at the inlet among various studies). Accordingly issues concerning the treatment of the outlet boundary condition, inlet turbulence parameters, optimal nodal configurations for a practical grid independent solution, appropriate turbulence model etc., have remained unanswered and too some extent ambiguous.

Consequently, the primary objective of the present work has been to configure a reliable and robust mathematical model for the hydrodynamic analysis of steelmaking...
tundish system. Towards these, the role of boundary conditions, inlet turbulence parameters, turbulence models and nodal configurations on the mathematical model predictions were investigated computationally. To demonstrate the present approach and to assess the computational results directly, experimentally measured RTD parameters from a water model tundish have been applied. The present work represents our first step towards developing a mathematical model concurrently with a computer aided measurement system for investigating RTD phenomena in a tundish.

2. Model Description

The flow produced in a twin strand steelmaking tundish by submerged stream is simulated through water flowing in a perspex glass tundish. Figure 1 shows the model tundish. The dimension of the tundish, submerged stream, and the water flow rate are obtained from geometrical and dynamical similarity conditions. Table 2 shows the range of model parameters and those of industrial tundish. The model tundish is designed to operate at 1.55 × 10^-3 m³/s water flow rate. This flow rate corresponds to an average velocity ($V_{in}$) of 0.493 m/s for a 20 mm inlet stream diameter.

2.1. Mathematical Model

A commercial CFD package FLUENT® is used to calculate the steady state velocity field produced in the tundish due to turbulent flow of an incompressible single-phase fluid under isothermal condition. The governing equations

| Boundary conditions | Numerical procedure |
|---------------------|---------------------|
| Inlet               | Outlet              | Grids applied (X,Y,Z) | Model configurations | No. of strands | Ref. |
| ladie outlet NA NA  | standard outflow    | 30x10x12             | steady, 3D, k-ε, ST  | 2              | 2.3 |
| ladie outlet 0.01V_{in}² | 2ν_{in}/D_{in}  | standard outflow    | 25x12x16             | steady, 3D, k-ε, ST | 2 | 4 |
| meniscus NA NA      | ambient pressure    | 25x15x15             | steady, 3D, k-ε, ST + magnetic | 1 | 5 |
| meniscus - -        | standard outflow    | 55x18x25             | steady, 3D, k-ε, ST  | 1 | 6 |
| ladie outlet NA NA  | standard outflow    | 40x20x15             | steady, 3D, k-ε, ST  | 2 | 7 |
| meniscus NA NA      | ambient pressure    | 25x11x15             | steady, 3D, k-ε, ST + magnetic | 2 | 9 |
| meniscus NA NA      | ambient pressure    | 25x18x15             | steady, 3D, k-ε, ST + magnetic | 2 | 10 |
| ladie outlet NA NA  | standard outflow    | 30x12x10             | steady, 3D, k-ε, ST  | 2 | 11 |
| port 0.005V_{in}²   | 1.94ν_{in}/A_{in}   | standard outflow    | 30x20x12             | steady, 3D, k-ε, ST | 2 | 12 |
| port 0.003 0.0003   | standard outflow    | NA                   | steady, 3D, k-ε, ST + nonisothermal | 1 | 13 |
| meniscus - -        | standard outflow    | NA                   | steady, 3D, k-ε, ST  | 2 | 14 |
| ladie outlet NA NA  | standard outflow    | 61x23x23             | steady, 3D, k-ε, ST  | 2 | 15 |
| port NA NA          | ambient pressure    | 45x34x27             | steady, 3D, k-ε, ST  | 2 | 16 |
| meniscus NA NA      | standard outflow    | 40x15x20             | steady, 3D, k-ε, ST  | 1, 2 | 18 |
| port 0.01V_{in}²    | 1.83ν_{in}/A_{in}   | standard outflow    | NA                   | steady, 3D, k-ε, ST + MP | 6 | 19 |
| port 0.01V_{in}²    | 1.83ν_{in}/A_{in}   | standard outflow    | 38,000               | steady, 3D, k-ε, ST + MP | 2 | 20,21 |
| ladie outlet NA NA  | standard outflow    | 80,000               | steady, 3D, k-ε, ST  | 4 | 22 |
| ladie outlet 0.003V_{in}² | 2.15ν_{in}/D_{in}  | standard outflow    | 1,80,000             | steady, 3D, k-ε, ST | 1 | 23 |
| ladie outlet NA NA  | standard outflow    | 2,86,000             | steady, 3D, k-ε, ST  | 1 | 24 |
| meniscus NA NA      | standard outflow    | 30x11x12             | steady, 3D, k-ε, ST + nonisothermal | 2 | 25 |
| ladie outlet NA NA  | standard outflow    | NA                   | steady, 3D, k-ε, ST  | 2 | 26 |
| NA 0.01V_{in}²      | 1.83ν_{in}/A_{in}   | standard outflow    | 39x37x14             | steady, 3D, k-ε, ST  | 1 | 28 |
| meniscus 0.01V_{in}² | 2ν_{in}/D_{in}     | standard outflow    | 40x17x16             | steady, 3D, k-ε, ST + nonisothermal | 1, 2 | 29,30 |
| meniscus NA NA      | ambient pressure    | NA                   | steady, 3D, two fluid, ST + magnetic | 1 | 31 |
| ladie outlet 0.01V_{in}² | 2ν_{in}/D_{in}   | standard outflow/ambient pressure 32,000 | steady, 3D, k-ε & Reynolds stress, ST (isothermal) | 2 | Present study |

| R_e, D_e, A_e: radius, diameter and cross-sectional area of the inlet nozzle respectively; NA not available; 3D three Dimensional; ST scalar transport (concentration/temperature); MP multi phase. |
of time-averaged flow and scalar transport for the Cartesian geometry are rather well known and consequently not reproduced here. From the predicted velocity field, C curve is obtained by solving a transient tracer dispersion equation. Semi Implicit Method for Pressure Linked Equation (SIMPLE) algorithm was used for pressure velocity coupling and first order upwind scheme for momentum and scalar transport equations. Figure 2 shows the numerical scheme used to determine C-curve in the present work. The convergence criterion for scaled residuals were set to be less than $10^{-3}$ except for concentration which is set to $10^{-5}$. Under relaxation was applied to pressure momentum and specific kinetic energy as well as its dissipation rate to seek the numerical solution. Under relaxation factor of 0.3, 0.7 and 0.8 respectively were applied on pressure, momentum and turbulence kinetic energy parameters. For a typical numerical calculation, standard values of the empirical constants of the turbulence model taken from the literature were applied. In the vicinity of the solid walls, the standard wall function was applied and at the solid wall, no slip for flow and zero flux for tracer transport were imposed. In addition, at the free surface zero shear stress condition was applied. The turbulence kinetic energy ($k_{in}$) and its dissipation rate ($\epsilon_{in}$) at the inlet were taken to be 0.0025 m$^2$/s$^2$ and 0.012 m$^2$/s$^3$ respectively.

In this way, the influence of the following parameters on the C curve is investigated to arrive at an optimal model configuration:

a) grid size: varied in between $3.2 \times 10^3$ to $1.5 \times 10^5$ cells
b) boundary condition at the tundish outlet: standard outflow, specified pressure
c) location of inlet boundary condition: ladle outlet, free surface and port region and
d) different turbulence model: Reynolds stress model, standard $k$-$\epsilon$ model

### Table 2. Characteristic parameters of industrial and model tundish

| Parameters                  | Industrial tundish | Model tundish |
|-----------------------------|-------------------|---------------|
| $L$, m                      | 3.0 - 8.0         | 1.0           |
| $W$                         | 0.10 - 0.30       | 0.23          |
| $H$                         | 0.10 - 0.30       | 0.26          |
| $L/L_{wall}$                | 0.68 - 0.62       | 0.69          |
| $\frac{d_{inlet} \cdot H}{L}$ | 0.08 - 0.82       | 0.15          |
| Side walls inclination      | 0 - 15"           | 10"           |
| Fr                          | 0.45 - 111        | 1.24          |
| Re                          | $10^2 - 10^3$     | $10^4$        |
| Volumetric flow rate, m/s   | $1.8 \times 10^{-3}$ - $2.8 \times 10^{-4}$ | $1.55 \times 10^{-4}$ |

2.2. Physical Modelling

Figure 3 shows the schematic view of the complete experimental system. It consisted of a model tundish, water supply and the instrumentation to record the tracer concentration at the exit of the tundish. From the storage tank water was transferred into a 0.5 m$^3$ cylindrical vessel. At the bottom of the vessel, a 250 mm long pipe (collector pipe) was attached via a gate valve. Tracer injection was done with the help of a pneumatically operated cylinder-piston assembly and a solenoid valve.

Self designed and calibrated platinum probes were installed at both the exits of the tundish (main probes), and in the cylindrical vessel (reference probe) to record the change in electrical conductivity of water due to tracer injection continuously on a desktop computer with the help of data acquisition card (DAS). A software was developed to coordinate the data collection and its processing. Data were stored into arrays and transferred into data files. These data were processed further to give a C curve.

Tundish was filled with water to the depth 0.26 m. After attaining steady state, 20 mL of potassium chloride was injected as a pulse into the inlet stream. The conductivity of water at both the exits and that of reference was recorded continuously on the PC for 20 min duration at a rate of 5 samples/s. The conductivity data were converted into concentration and thereby, the C-curve was obtained. In each experiment, mass balance of tracer was made and output of the tracer lie in between 90–96% of the input.

3. Results and Discussion

Many experimental and computational studies on steel-making tundish system has been reported in the literature. A summary of these is presented in Table 1 to demonstrate the extent of variations in these studies in terms of the modelling methodology. It is at once apparent from Table 1 widely varying tundish shapes and geometries were mathematically modelled embodying numerical procedures, grid configurations and boundary conditions that varied considerably from one study to another. For example, the location of the inflow boundary plane, the types of outlet boundary conditions, the distribution of the turbulence parameters at the inlet, the number of grids in the computational domain etc. varied considerably among studies considered in Table 1. Since numerical results are sensitive to these, consequently major conclusions drawn from various studies have appeared to become different. To illustrate this point better
Figs. 4(a) and 4(b) have been included in the text, in which, comparison between experimental and predicted tracer response characteristics in a tundish as reported by two different groups of investigators have been shown. There, while one set of results indicate excellent agreement, the other in contrast, suggests serious differences between experimental data and predicted results. A systematic analysis is therefore required to resolve such apparent anomaly reported within the literature. Despite many studies till date, an optimal hydrodynamic model remains yet to be configured for steelmaking tundish system.

Several experiments are performed to determine C curves for both strands of a tundish without any flow modifiers. C curves were found to be practically similar for both strands of the tundish. As an example, Fig. 5 shows experimental C curves for both strands of the tundish. Both the C curves can be seen to be similar in their respective nature of variation. Each curve is characterized by two peak values of concentration, one soon after the tracer injection and the other after some time. After attaining the second peak, tracer concentration decays with time. Appearance of two peak values of concentration in both the curves suggests short-circuiting in the tundish fluid flow system. Short-circuiting of the fluid is an undesirable feature.
Based on the above experimental observations, extensive numerical calculations are made to develop a mathematical model, which can describe the observed variation in concentration with time at both strands of the tundish.

During a typical flow calculation, it was observed that computed inlet mass flow rate was extremely sensitive to grid distribution at the inlet. Consequently, the inlet grid was refined till the computed mass flow rate approximately equals to the prescribed one. In Fig. 6 the dimensionless mass flow rate (computed/prescribed) is plotted against the reciprocal of unit cell volume size applied. As seen from the figure, the computed profile changes sharply up to a cell volume of 3.4 mm$^3$, moderately beyond that and becomes practically equal to the prescribed value beyond a size of 1.7 mm$^3$ of a unit cell. On the basis of above observation, it was decided to use at least a size of 1.7 mm$^3$ of a unit control volume at the inlet (and hence at the outlet) for all further calculation reported in this study.

In the next phase of computational work, influence of tundish grid size, types of turbulence model, and boundary conditions at the various associated boundaries on the computed C curve is studied.

3.1. Effect of Tundish Grid Size

Three sets of grid with total number of volume elements of 32 000; 51 000; and 150 000 were generated via the Geometric and Mesh Building Interactive Tool (GAMBIT) of FLUENT package to study the effect of tundish grid size on predicted C curve. These volume elements correspond to $31 \times 10^{-6}$, $12 \times 10^{-6}$ and $6 \times 10^{-6}$ dimensionless control volume of the present study. Figure 7 shows the predicted C curves for the different grid configurations. In these computations, the standard $k$-$\varepsilon$ model was applied. A specified velocity was applied at the ladle outlet while the standard outflow boundary condition at the exit nozzle was considered. As seen, of all the three computed C curves, the ones deduced via 51 000 and 150 000 cells (corresponding to an average unit cell volume of 1 450 to 490 mm$^3$ respectively) are relatively more close. From the nature of the predicted results, one might anticipate here that deploying more than 150 000 cells in the computational domain is unlikely to produce any significant variation on the nature of the predicted C curve and therefore minimal benefit in terms of improvement in accuracy of the predicted results.

Consequently, results obtained via 1 500 000 cells can be essentially taken to be grid independent. It is because of such that results produced via 51 000 cells have been considered to be physically more meaningful than those deduced via 51 000 cells despite the latter having somewhat close correspondence to the experimental curves presented in Fig. 5. In view of such, it has been decided to apply 150 000 cells (corresponding to $6 \times 10^{-6}$ dimensionless control volume) in the computational domain so that predicted results can be considered to be essentially independent of nodal configurations. It is interesting to note from Table 1 that in majority of the studies reported till date, considerably sparse grid systems or smaller number of nodes than is advocated in the present study were deployed.

3.2. Effect of Outlet Boundary Condition

The tundish exit can be computationally treated as (i) a standard outflow or (ii) as a plane or surface, at which flow occurs at an ambient pressure (taken). Figure 8 shows the
predicted C curves for the above two boundary conditions. These indicate that the 2nd peak in the tracer dispersion curve do not appear at all when a "specified pressure" boundary condition is invoked at the outlet. In contrast, the "two peak characteristics" of the measured tracer dispersion curve is predicted reasonably well when the standard outflow boundary condition at the exit is applied. To illustrate the reason for such differences in the predicted results further, mass flow rates through the tundish outlet were computed and compared with the inflow rate. This is shown in Fig. 9 for two different types of outlet boundary conditions mentioned already. There, it is readily apparent that the standard outflow type boundary condition at the tundish outlet leads to relatively more accurate overall mass balances in the system (in comparison to those deduced via a specified ambient pressure condition at the outlet boundary). As a consequence, it was decided to carry out all future numerical computations by prescribing the standard outflow boundary conditions at the tundish nozzle outlet.

3.3. Effect of Location of Inlet Boundary

In most of the computational studies, the location of the inlet boundary was chosen at the port/meniscus region. Few authors have applied ladle outlet also. It is apparent that prescription of a flat velocity profile at the port/meniscus region is not a suitable representation of the actual velocity profile. The frictional forces acting on the flowing fluid by the walls of the nozzle is likely to alter the velocity profile; thus ladle outlet appears to be a realistic location of the inlet boundary. This can be confirmed from the computational results shown in Fig. 10, in which predicted axial velocity along the centre-line of the ladle shroud is shown. There the axial velocity at the ladle port is seen to be somewhat higher than those applied at the ladle outlet (constant area average velocity). As a consequence of such, one might anticipate that the velocity profile at the port region is different than a constant, flat and uniform velocity profile. To computationally investigate the implications of such trend of results on the overall tundish hydrodynamics, C curves were predicted considering three different locations of the inlet boundary:

- the port
- the meniscus and
- the ladle outlet

Figure 11 shows the computed C curves for the above conditions. As expected the concentration profile obtained by considering the ladle outlet location is different from the remaining ones. It can be seen in the figure that inlet boundaries applied either at port or meniscus lead to nearly identical concentration profiles. Following this it was decided to consider ladle outlet location as the inlet boundary for tundish flow calculations.

3.4. Optimal Configuration

In the numerical results presented so far, we simulated a bare tundish in order to establish a reliable mathematical
model to study fluid flow phenomena in tundish. Thus applying a relatively grid size (>150,000 volume elements), standard coefficient $k$–$\varepsilon$ turbulence model, a standard outflow conditions at the tundish outlet and selecting an inlet boundary at the ladle outlet, C curves were predicted for a tundish fitted with a dam. In Fig. 12 comparison between experimental measurements and numerical prediction is finally presented. Evidently, as seen, there is close correspondence between model predictions and experimental measurements. This confirms that the mathematical model developed in this study is sufficiently robust and reliable for numerical simulation of flow phenomena in steelmaking tundish systems.

3.5. Role of Turbulence Model

In the hydrodynamic modelling of tundish system $k$–$\varepsilon$ turbulence model is extensively used by many researchers (see Table 1). Flow in the tundish is fairly complicated. It is highly turbulent near the inlet region and weakly turbulent near the side walls. As a result $k$–$\varepsilon$ turbulence model which is valid for high Reynolds number flow may not be strictly applicable to the entire tundish domain. It contrasts to the $k$–$\varepsilon$ model, Reynolds stress model (RSM)\(^{33}\) does not have such limitations as it directly solves the relevant Reynolds stress equations. Figure 13 shows the predicted C curves for standard $k$–$\varepsilon$ model (solid line) and Reynolds stress model (dashed line). In these, the standard outflow boundary condition was applied at the tundish exit while a known velocity (deduced from the mass inflow rate at inlet nozzle dimension) was applied at ladle outlet. It is interesting to note that the times to attain peak concentration are nearly identical for both the models. It may be noted that RSM takes though higher CPU times for calculation as compare to $k$–$\varepsilon$ turbulence model but predicts C curves close to the experiments. Only one other\(^{24}\) investigation has reported on the accuracy of RSM. However, further work is needed. As a final point, in the present work, a steady state flow situation has been modelled numerically for which standard coefficient $k$–$\varepsilon$ turbulence model has been found to be adequate. However, for modelling more challenging flow situations (primarily transient phenomena such as, ladle change operation etc.) LES (Large Eddy Simulation) is expected to be relatively more effective than the $k$–$\varepsilon$ turbulence model. LES is becoming popular among researchers in metals processing and has been applied recently, to un-score the time dependent flow phenomena in the mold region of a continuous casting machine.

4. Conclusions

A combined experimental and computational study was carried out to develop a mathematical model for investigating tundish flow phenomena. From the present study the following conclusions can be drawn:

(1) Predicted results are found to be sensitive to grid distribution. A very dense grid within the inlet nozzle and sufficiently fine grid in the bulk of the tundish is required to obtain practically grid independent solutions.

(2) Although Reynolds stress model is somewhat superior to $k$–$\varepsilon$ turbulence model, the former extremely CPU intensive. Consequently, from the viewpoint of optimal accuracy and CPU time, $k$–$\varepsilon$ turbulence model can be incorporated in the mathematical model for fluid flow analysis of turbulent tundish.

(3) The treatment of inlet and outlet boundary conditions was found to have some influence on the predicted results. For our study a standard outflow type of boundary condition at the exit and the specified inlet velocity at the ladle outlet (upstream of the bath surface) were found to be
adequate.

(4) A close similarity between experimental and predicted C-curves was observed with and without flow modifiers. Several numerical simulations carried out in this work indicated that the mathematical model developed was sufficiently reliable and robust.

REFERENCES
1) D. Mazumdar and R. I. L. Guthrie: ISIJ Int., 39 (1999), 524.
2) S. Chakraborty and Y. Sahai: Metall. Trans. B, 23B (1992), 153.
3) Y. Sahai and T. Emi: ISIJ Int., 36 (1996), 1166.
4) K. M. Lai, M. Salcudean, S. Tanaka and R. I. L. Guthrie: Metall. Trans. B, 17B (1986), 449.
5) O. J. Ilegbusi and J. Szekely: Ironmaking Steelmaking, 16 (1989), 110.
6) M. C. Tasi and M. J. Green: Steelmaking Conf. Proc., ISS, Warrendale, PA, (1991), 501.
7) S. Chakraborty and Y. Sahai: ISIJ Int., 31 (1991), 960.
8) S. Chakraborty and Y. Sahai: Metall. Trans. B, 22B (1991), 429.
9) O. J. Ilegbusi and J. Szekely: Ironmaking Steelmaking, 16 (1989), 110.
10) O. J. Ilegbusi and J. Szekely: Steel Res., 59 (1988), 399.
11) Y. He and Y. Sahai: Metall. Trans. B, 18B (1987), 81.
12) K. H. Tacke and J. C. Ludwig: Steel Res., 58 (1987), 262.
13) Y. Miki and B. G. Thomas: Metall. Trans. B, 30B (1999), 639.
14) S. Lopez-Ramirez, J. Palafoux-Ramos, R. D. Morales, M. A. Barron-Meza and M. V. Toledo: Steel Res., 69 (1998), 423.
15) H. S. Chen and R. D. Pehlke: Metall. Trans. B, 27B (1996), 745.
16) C. M. Fan, S. M. Pan, H. S. Wang and W. S. Hwang: Ironmaking Steelmaking, 29 (2002), 376.
17) P. K. Jha, S. K. Dash and S. Kumar: Ironmaking Steelmaking, 29 (2002), 36.
18) A. Robert and D. Mazumdar: Steel Res., 72 (2001), 97.
19) L. Garcia-Demecides, R. D. Morales, S. Lopez-Ramirez, J. de J. Barreto-Sandoval, J. Palafoux-Ramos and M. Diaz-Cruz: Steel Res., 72 (2001), 346.
20) R. D. Morales, M. Diaz-Cruz, J. Palafoux-Ramos, S. Lopez-Ramirez and J. de J. Barreto-Sandoval: Steel Res., 72 (2001), 81.
21) R. D. Morales, J. de J. Barreto, S. Lopez-Ramirez, J. Palafoux-Ramos and D. Zacharias: Metall. Trans. B, 31B (2000), 1505.
22) R. D. Morales, S. Lopez-Ramirez, J. de J. Barreto and J. Palafoux-Ramos: Steelmaking Conf. Proc., ISSN, Warrendale, PA, (2000), 955.
23) K. J. Craig, D. J. de Kock, K. W. Makgata and G. J. de Wet: ISIJ Int., 41 (2001), 1194.
24) H. J. Odenthal, H. Pfeifer and M. Klaas: Steel Res., 71 (2000), 210.
25) C. S. Damle and Y. Sahai: ISIJ Int., 36 (1996), 681.
26) H. Chen and R. D. Pehlke: Steelmaking Conf. Proc., ISSN, Warrendale, PA, 1994, 695.
27) C. S. Damle and Y. Sahai: Steelmaking Conf. Proc., ISSN, Warrendale, PA, 1994, 235.
28) F. Shen, J. M. Khodadadi, S. J. Pien and X. K. Lan: Metall. Trans. B, 25B (1994), 669.
29) S. Joo and R. I. L. Guthrie: Metall. Trans. B, 24B (1993), 755.
30) S. Joo, J. W. Han and R. I. L. Guthrie: Metall. Trans. B, 24B (1993), 779.
31) O. J. Ilegbasi: ISIJ Int., 34 (1994), 732.
32) B. E. Launder and D. B. Spalding: Computer Methods in Applied Mechanics and Eng., 3 (1974), 269.
33) W. Rodi: Turbulence models and their application in hydraulics: A state of the art review, Internal Report, University of Karlsruhe, Germany, (1988), 33.