Numerical Study on Steel Flow and Inclusion Behavior during a Ladle Teeming Process

Peiyuan Ni

Licentiate Thesis

Stockholm 2013

Division of Applied Process Metallurgy
Department of Materials Science and Engineering
School of Industrial Engineering and Management
Royal Institute of Technology
SE-100 44 Stockholm
Sweden

Akademisk avhandling som med tillstånd av Kungliga Tekniska Högskolan i Stockholm, framlägges för offentlig granskning för avläggande av Teknologie Licentiatexamen, söndagen den 21 mars, kl. 09.30 i Sefström (M131), Brinellvägen 23, Materialvetenskap, Kungliga Tekniska Högskolan, Stockholm

ISRN KTH/MSE--13/05--SE+APRMETU/AVH
Peiyuan Ni    Numerical Study on Steel Flow and Inclusion Behavior during a Ladle Teeming process

Division of Applied Process Metallurgy
Department of Materials Science and Engineering
School of Industrial Engineering and Management
Royal Institute of Technology
SE-100 44 Stockholm
Sweden

ISRN KTH/MSE--13/05--SE+APRMETU/AVH

© Peiyuan Ni, February, 2013
To my beloved parents
Abstract

Inclusions in molten steel have received worldwide concern due to their serious influence on both the steel product quality and the steel production process. These inclusions may come from the deoxidation process, reoxidation by air and/or slag due to an entrainment during steel transfer, and so on. They can break up a casting process by clogging a nozzle. A good knowledge on both steel flow and inclusion behavior is really important to understand nozzle clogging, as well as to take some possible measures to alleviate clogging. In this thesis, steel flow and inclusion behavior during a teeming process were investigated by mathematical simulations with verification by pilot-plant experiments.

Firstly, steel flow phenomena during a ladle teeming process were studied. Different turbulence models, including the low Reynolds number $k-\varepsilon$ model and the realizable $k-\varepsilon$ model both with an enhanced wall treatment (EWT) and a standard wall function (SWF), were used to simulate this process. All of these turbulence model predictions generally agreed well with the experimental results. The velocity distributions in the nozzle were also predicted by these turbulence models. A large difference of the boundary-layer velocity predicted with these two near wall treatment methods was found. At the late stage of the teeming process, the drain sink flow phenomena were studied. The combination of an inclined ladle bottom and a gradually expanding nozzle was found to be an effective way to alleviate a drain sink flow during teeming.

Then, inclusion behavior during a teeming stage was studied. A Lagrangian method was used to track the inclusions in steel flow and compare the behaviors of different-size inclusions. In addition, a statistical analysis was conducted by the use of a stochastic turbulence model to investigate the behaviors of different-size inclusions in different nozzle regions. Inclusions with a diameter smaller than 20$\mu$m were found to have a similar trajectory and velocity distribution in the nozzle. However, inertia force and buoyancy force were found to play an important role for the behavior of large-size inclusions or clusters. The statistical analysis results indicate that the regions close to the connection between different angled nozzle parts seem to be very sensitive for an inclusion deposition.

Key words: steel flow, ladle teeming, numerical simulation, inclusion behavior, CFD, clogging, deposition.
Acknowledgements

First of all, I would like to express my deepest gratitude and appreciation to my excellent supervisors Professor Lage Jonsson, Professor Pär Jönsson and Doctor Mikael Ersson for your professional guidance, great encouragement and patience.

I would like to thank Doctor Anders Tilliander for your kind help in my work at KTH.

Many thanks to Professor Sichen Du at KTH for your kind help and care for my study at KTH.

All my colleagues in the division of TPM and all my friends in Sweden are thanked for their kind help and good friendship. All the happy times of eating delicious food, doing exercises and traveling with you make my life in Sweden colorful and interesting. Special thanks to Mr. Erik Roos for your great help in my work.

I also appreciate Professor Ting-an Zhang, associate Professor Zhihe Dou, Doctor Guozhi Lv and other people in the research team of Northeastern University in China. Thanks for your help during my four-year study life in your team.

The Financial support by the China Scholarship Council is acknowledged.

Also, I would like to thank my parents who give me endless love and support.

Peiyuan Ni
Stockholm, January 2013.
Supplements

The present thesis is based on the following supplements:

Supplement 1
Simulations of the Ladle Teeming Process and Verification with Pilot Experiment
Peiyuan Ni, Lage T.I. Jonsson, and Pär G. Jönsson
Accepted for publication in Steel Research International.

Supplement 2
Turbulent Flow Phenomena and Ce₂O₃ Behavior during a Steel Teeming Process
Peiyuan Ni, Lage T.I. Jonsson, Mikael Ersson and Pär G. Jönsson
Accepted for publication in ISIJ International.

The contributions by the author to the supplements of this thesis:

Supplement 1
Literature survey, numerical simulation and major part of writing

Supplement 2
Literature survey and major part of numerical simulation and writing
Chapter 1 Overview

1.1 Introduction

Inclusions in molten steel have received worldwide concern due to their serious influence on both the steel product quality and the steel production process. This causes steelmakers to stringently control the steelmaking process parameters, especially the cleanness of molten steel. These inclusions may come from the de-oxidation process, re-oxidation by air and/or slag due to an entrainment during steel transfer, and so on. The influence of inclusions on a continuous casting process is quite serious. They can break up a casting process by clogging a tundish nozzle when they pass close to an inside nozzle wall and deposit onto the wall. Solidification of steel on the nozzle wall due to a too low steel temperature will accelerate the clogging process. In addition, inclusions can lead to quality problems in the final steel product.

During a ladle teeming process, some steel flow phenomena, like the drain sink flow and the vortex drain sink flow, can lead to that slag, which is floating on the top surface of steel, is entrained into steel. Oxides of entrained slag may directly become inclusions in steel or react with strong deoxidizers in steel under the formation of inclusions. Once a slag entrainment occurs, the teeming process is then interrupted and a lot of steel will be left in the ladle. This will decrease the productivity. In addition, the steel temperature also changes due to heat losses during teeming. Owing to the important influence of a ladle teeming on steel cleanness, steel productivity and steel temperature, it received a great deal of research efforts[1–20]. A drain sink flow at the late stage of teeming is one of the reasons for slag entrainment. The reason for a drain sink flow is that the volume flow of molten steel at the ladle bottom becomes smaller than the corresponding orifice capacity.[7] Previous researches[3,7,10,12] showed that the critical drainage height was dependent on the outlet diameter, but little related to the outlet location and the ladle diameter. Theoretical models[1,3,7,10] were also developed to predict the critical drainage height. Very few papers studied the drain sink flow by using mathematical modeling. Kojola et al.[18] developed a mathematical model which considered the effect of a supernatant phase to predict the drain sink. Mazzaferro et al.[12] modeled the drain sink
flow in a ladle with an inclined bottom geometry. The modeling result showed that the critical drainage height depended on the nozzle diameter. It also showed that the inclination of a ladle bottom can strongly reduce the steel left in a ladle.

When inclusions are present in steel, they may lead to nozzle clogging. Some research has been carried out to remove inclusions in molten steel during a ladle treatment as well as during a tundish operation.\textsuperscript{[21-24]} However, it is impossible to obtain a completely clean steel with the current steel production technology. Clogging is also closely related to the inclusion behavior in the steel. Therefore, the knowledge on the steel flow and the inclusion behavior is rather important for an understanding of the nozzle clogging process and for making a prediction on clogging situations. A great amount of mathematical simulations have propelled our understanding of the reality of both the steel flow and inclusion behavior.\textsuperscript{[25-47]} As early as 1973, Szekely et al.\textsuperscript{[25]} modeled fluid flow in a mold with a straight nozzle and a bifurcated nozzle, respectively. In addition, Thomas and Bai et al.\textsuperscript{[27-35]} carried out systematic research on steel flows in nozzles, which investigated the effects of nozzle parameters and operating practice on the steel flow characteristics in a submerged entry nozzle (SEN). Furthermore, steel flows in nozzles were also studied by using different turbulence models\textsuperscript{[42-47]}. Some researchers also studied the inclusion behavior in a nozzle during casting. Wilson et al.\textsuperscript{[26]} investigated the steel flow characteristics in a nozzle and tracked the trajectories of inclusions. The deposition of inclusions due to a centripetal force and turbulence was also studied. Yuan\textsuperscript{[36]} et al. predicted the fraction of inclusions with different densities and sizes trapped by a lining in a stopper-rod nozzle. Zhang et al.\textsuperscript{[37-39]} tracked the trajectories and entrapment locations of inclusions in a slide-gate nozzle. Long et al.\textsuperscript{[47]} studied the Al$_2$O$_3$ inclusion behavior in a turbulent pipe flow. Effects of factors, such as release location of inclusion, inclusion size, pipe diameter, casting speed, on the entrapment probability were investigated.

1.2 Aim of the Thesis

Due to the important influence of steel flow and inclusions on steel production process and steel product quality, steel flow phenomena and inclusion behavior during a ladle teeming process were studied in this thesis by using mathematical simulations. As is
summarized, previous research did not consider the influence of nozzle geometry on the drain-sink-flow phenomena enough. Steel flow properties in boundary layer near a nozzle wall are not completely understood. Research on inclusion behavior coupled with molten steel flow in a nozzle is far from enough. Behaviors of different kinds of inclusions with different sizes released from different locations of a nozzle are not fully understood. In order to make a good understanding on both the steel flow and the inclusion behavior in nozzle, simulations with experimental verifications were carried out in the following two supplements.

**Supplement 1**
- Firstly, steel flow phenomena, especially in nozzle, during teeming process were studied. The realizable $k$-$\varepsilon$ model with different wall treatments, a standard wall function (SWF) method and an enhanced wall treatment (EWT) method, and the low Reynolds number $k$-$\varepsilon$ model were used to simulate the steel flow. The steel flow velocity distributions in nozzle predicted by experimental verified models were compared. At the late stage of teeming, the influence of nozzle geometry and ladle bottom geometry on the drain sink flow phenomena was investigated.

**Supplement 2**
- Then, turbulence properties of steel flow and Ce$_2$O$_3$ inclusion behavior in nozzle were studied. Ce$_2$O$_3$ is difficult to remove due to that its density is close to steel. Furthermore, it tends to stick on the nozzle wall when it comes close to the wall. Behaviors of different-size Ce$_2$O$_3$ inclusions released from different locations were studied at a teeming stage. Effect of the stochastic turbulence on inclusions of different sizes was also studied. In addition, a statistical analysis was carried out to investigate the inclusion behavior at different nozzle regions.
Chapter 2 Experimental Verification

The simulation work was based on the following experiment. And, the experimental results were used to verify the developed models in this thesis.

Steel was melted in a 600 Hz induction furnace (Figure 1) of a 0.48 m inner diameter and with an Al$_2$O$_3$-lining, a 600 kg nominal melt size and an 800 kVA electrical output power. At the bottom of the furnace, a zirconia nozzle with a diameter of 5 mm was installed. The steel temperature was monitored continuously by two Rh-Pt thermocouples. After a total melting of the steel and an adjustment of the temperature and compositions to the desired experimental level, an alumina-graphite stopper rod was elevated and the steel flowed through the zirconia nozzle with a controlled temperature and into a mold placed on a scale. The cumulative mass on the scale was sampled by a 1 Hz frequency. Then, the change of the cumulative weight of steel in mold with time could be monitored.

Figure 1. Scheme of the experimental equipment. [52]
Chapter 3 Mathematical Model Descriptions

3.1 Governing Equation

The governing equations for the conservation of mass and momentum for a transient state and an incompressible fluid flow are as follows:

- Continuity Equation:
  \[
  \frac{\partial}{\partial x_i} (\rho u_i) = 0
  \]  

- Momentum Equation:
  \[
  \frac{\partial}{\partial t} \rho u_i + \frac{\partial}{\partial x_j} (\rho u_i u_j) = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \rho g_i
  \]

where \(\rho\) is the density, \(\mu\) is the molecular viscosity, \(\mu_t\) is the turbulence viscosity and \(g_i\) is the gravity in \(i\) direction.

3.2 Turbulence Models

3.2.1 The realizable \(k-\varepsilon\) turbulence model

The realizable \(k-\varepsilon\) model in FLUENT\(^{[53]}\) is used to calculate the turbulence viscosity:

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}
\]  

where \(C_\mu\) is a constant, \(k\) is the turbulence kinetic energy and \(\varepsilon\) is the turbulence dissipation rate. The equations of the \(k-\varepsilon\) turbulence model for calculating the turbulence kinetic energy and dissipation rate are given below:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_i} \left[ (\mu + \mu_t) \frac{\partial k}{\partial x_i} \right] + \frac{\partial}{\partial x_i} \left[ (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial \varepsilon}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_i} \left[ (\mu + \frac{\mu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b + \rho C_1 \varepsilon \sigma_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\varepsilon}}
\]

where \(C_\mu = 0.09\), \(C_{1\varepsilon} = 1.44\), \(C_2 = 1.9\), \(\sigma_k = 1.0\) and \(\sigma_\varepsilon = 1.2\) are constants in the turbulence model.
3.2.2 The low Reynolds number \( k-\varepsilon \) turbulence model

The low Reynolds number \( k-\varepsilon \) model\(^{[53-55]}\) from Change, Hsieh and Chen were used to simulate the teeming process. It can be used to make turbulence model calculations to a solid wall where low Reynolds number flow exists. The turbulence quantities \( k \) and \( \varepsilon \) are determined from the following transport equations:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \rho \varepsilon \tag{6}
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + f_1 C_1 \mu_t \varepsilon \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \rho f_2 C_2 \frac{\varepsilon^2}{k} + E \tag{7}
\]

The empirical turbulence model constants and damping functions are listed in Table 1.

**Table 1.** Summary of model constants and functions in governing equations.

<table>
<thead>
<tr>
<th>Constant</th>
<th>Value</th>
<th>( D )</th>
<th>( C_p )</th>
<th>( C_1 )</th>
<th>( C_2 )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\varepsilon )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Function</td>
<td>( \bar{\varepsilon} )</td>
<td>( \bar{\varepsilon}_w )</td>
<td>( \mu_t )</td>
<td>( \mu )</td>
<td>( Re_T )</td>
<td>( Re_y )</td>
<td>( Re_\varepsilon )</td>
</tr>
<tr>
<td>Expression</td>
<td>( \varepsilon + D )</td>
<td>( \varepsilon_w = \mu \frac{\partial^2 k}{\partial y^2} )</td>
<td>( \rho f_1 C_1 \frac{k^2}{\bar{\varepsilon}} )</td>
<td>( \rho \mu )</td>
<td>( \frac{\sqrt{\bar{\varepsilon}}}{\mu} )</td>
<td>( \frac{(\mu \varepsilon / \rho)^{1/4}}{\mu} )</td>
<td></td>
</tr>
<tr>
<td>Constant or Function</td>
<td>( E )</td>
<td>( f_1 )</td>
<td>( f_2 )</td>
<td>( f_\mu )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Value or Expression</td>
<td>0</td>
<td>1.0</td>
<td>( [1 - \exp(-0.0215 Re_y)]^2 )</td>
<td>( [1 - 0.01 \exp(-Re_\varepsilon^2)] )</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

3.2.3 The Kim-Chen modified \( k-\varepsilon \) turbulent model

The \( k-\varepsilon \) model by Chen and Kim\(^{[56]}\) has also been used to calculate the turbulence viscosity:

\[
\mu_t = \rho C_d C_\mu \frac{k^2}{\bar{\varepsilon}} \tag{8}
\]

where \( C_\mu \) and \( C_d \) are constants, \( k \) is the turbulence kinetic energy and \( \varepsilon \) is the turbulence dissipation rate. The equations of the \( k-\varepsilon \) turbulence model for calculating the turbulence kinetic energy and dissipation rate are given below:

\[
\rho \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} \left[ \rho k u_i - \frac{\mu_t}{\rhoRT(k)} \frac{\partial k}{\partial x_i} \right] = \rho (P_k + I_b - \varepsilon) \tag{9}
\]

\[
\rho \frac{\partial}{\partial t} \varepsilon + \frac{\partial}{\partial x_i} \left[ \rho \varepsilon u_i - \frac{\mu_t}{\rhoRT(\varepsilon)} \frac{\partial \varepsilon}{\partial x_i} \right] = \frac{\varepsilon}{\rho} \left( C_{1\varepsilon} P_k + C_{3\varepsilon} I_b + C_{2\varepsilon} \varepsilon \right) - \rho C_{\varepsilon\varepsilon} \frac{P_k^2}{k} \tag{10}
\]
where $C_{\mu} = 0.5478$, $C_d = 0.1643$, $C_{1\varepsilon} = 1.15$, $C_{2\varepsilon} = 1.9$, $C_{3\varepsilon} = 1.0$, $C_{4\varepsilon} = 0.25$, PRT$(k) = 0.75$, PRT$(\varepsilon) = 1.15$ are constants in the turbulence model. The parameters PRT$(k)$ and PRT$(\varepsilon)$ are the turbulent Prandtl number of variable $k$ and $\varepsilon$, respectively.

### 3.3 Interface Tracking Model

The Volume of Fluid (VOF) model was used in ANSYS FLUENT\[53\] to track the interface of gas and liquid. The volume fraction of gas phase and liquid phase is computed based on the following constraint:

\[ \alpha_l + \alpha_g = 1 \]  \hspace{1cm} (11)

where $\alpha_l$ is equal to 0 for a cell with steel volume fraction smaller than $1e^{-6}$ and 1 for a cell with steel volume fraction larger than $9.99999e^{-1}$. When the cell is located at the interface between steel and gas, $\alpha_l$ should be between 0 and 1. Also, the properties (e.g., viscosity) are computed in the following manner:

\[ F = \alpha_l F_l + (1 - \alpha_l) F_g \]  \hspace{1cm} (12)

The Height-of-Liquid model in Phoenics\[57\] was used to track the interface of gas and liquid. The position of the interface is determined by calculating the volume fraction of liquid, $\alpha_l$, in each cell from:

\[ \alpha_l = (\sum m - \sum m_1) / (\rho_l * V) \]  \hspace{1cm} (13)

The interface is defined to lie in the cells with an $\alpha_l$ value in the range: $0 < \alpha_l < 1$.

The density field can be obtained as follows:

\[ \rho = \rho_l * \alpha_l + \rho_g * (1 - \alpha_l) \]  \hspace{1cm} (14)

The viscosity field is calculated as follows:

\[ \mu = \mu_l * \alpha_l + \mu_g * (1 - \alpha_l) \]  \hspace{1cm} (15)

where $\rho_l$ is the liquid density, $\rho_g$ is the gas density, $\alpha_l$ is the volume fraction of liquid, $V$ is the cell volume, $\sum m$ is the total mass of liquid in a vertical column of cells, $\sum m_1$ is the total mass of liquid below the interface cells.

### 3.4 Particle Tracking Model

The particle tracking is done by using a Lagrangian method\[58\]. The positions of particles can be obtained by solving the following equation:
\[ \frac{dx_{pi}}{dt} = u_{pi} \]  

(16)

where \( x_{pi} \) is the particle position, \( u_{pi} \) is the particle velocity in \( i \) direction.

The particle velocity is obtained by solving the following particle momentum equation:

\[ m_p \frac{du_{pi}}{dt} = D_p (U_i - u_{pi}) + m_p g_i \left( 1 - \frac{\rho}{\rho_p} \right) \]  

(17)

where \( m_p \) is the mass of the particle, \( D_p \) is a drag function, \( U_i \) is the continuous-phase velocity, \( \rho_p \) is the particle density.

### 3.5 Particle Stochastic Turbulence Model

In order to incorporate the effect of turbulent fluctuations on inclusion motion, a stochastic turbulent model can be used. The first approach that was based on the eddy lifetime\cite{59} spawned the eddy-interaction models in which fluid velocities (eddies) are taken to be stochastic quantities, which remains constant for the lifetime of the eddy or, if shorter, the transit time of the particle through the eddy.\cite{60,61} The continuous-phase velocity can be expressed by the following equation:

\[ U_i = u_i + \dot{u}_i \]  

(18)

where \( u_i \) and \( \dot{u}_i \) are the continuous-phase average velocity and the fluctuating component, respectively.

### 3.6 Boundary Conditions

![Figure 2. Schematic diagram of: (a) calculation domain and inclusion release location and (b) nozzle regions.](image)
The computational domain can be seen in Figure 2, where inclusions are released from the curve with a radius 5 cm. The exact release location information is shown in Table 2. The boundary conditions of steel flow and inclusion motion are given in Table 3.

### Table 2. Release locations of inclusions.

<table>
<thead>
<tr>
<th>Locations</th>
<th>$\alpha$, defined in Figure 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5º</td>
</tr>
<tr>
<td>2</td>
<td>25º</td>
</tr>
<tr>
<td>3</td>
<td>45º</td>
</tr>
<tr>
<td>4</td>
<td>65º</td>
</tr>
<tr>
<td>5</td>
<td>85º</td>
</tr>
</tbody>
</table>

### Table 3. Summary of boundary conditions.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>$u$</th>
<th>$v$</th>
<th>$w$(3D)</th>
<th>$k$</th>
<th>$\varepsilon$</th>
<th>$P_{\text{gage}}$ (2D)</th>
<th>$I$, intensity(2D)</th>
<th>$D$, m</th>
<th>Inclusion</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>$\frac{\partial k}{\partial z} = 0$</td>
<td>$\frac{\partial \varepsilon}{\partial z} = 0$</td>
<td>0</td>
<td>1%</td>
<td>0.48</td>
<td>-</td>
</tr>
<tr>
<td>Outlet</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>$\frac{\partial k}{\partial z} = 0$</td>
<td>$\frac{\partial \varepsilon}{\partial z} = 0$</td>
<td>0</td>
<td>5%</td>
<td>0.005</td>
<td>escape</td>
</tr>
<tr>
<td>Wall</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>$\frac{\partial k}{\partial z} = 0$</td>
<td>$\frac{\partial \varepsilon}{\partial z} = 0$</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>stick</td>
</tr>
<tr>
<td>Axis(2D)</td>
<td>$\frac{\partial u}{\partial y} = 0$</td>
<td>0</td>
<td>0</td>
<td>$\frac{\partial k}{\partial y} = 0$</td>
<td>$\frac{\partial \varepsilon}{\partial y} = 0$</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

#### 3.7 Solution Methods

In the first paper, the geometry and the mesh of the ladle were done by ANSYSWORKBENCH 13.0\textsuperscript{[56]}. The commercial software ANSYS FLUENT 13.0\textsuperscript{[56]} was used for the calculations. The SIMPLE scheme was used for the pressure-velocity coupling. In addition, the PRESTO discretization method was adopted to discretize the pressure. The governing equations were discretized by using a second order upwind interpolation scheme. The convergence criteria were specified as follows: the residuals of all dependent variables must be less than $10^{-3}$.

In the second paper, the commercial software PHOENICS\textsuperscript{[57]} was used to solve the equations. A three-dimensional model of the ladle was developed. The Kim-Chen modified $k$-$\varepsilon$ model\textsuperscript{[55]} was used to describe the turbulence properties of steel flow. The Height-of-Liquid model, which is an option available in PHOENICS, was used to capture the moving interface of steel and air. The Log-Law wall function was used to bridge the near-wall layer and the outside fully developed turbulent flow region. Hybrid was used as the differencing scheme. The time step for the calculation of the steel flow was set as 0.001 s considering the calculation stability. The global convergence criterion in this calculation was set to 0.01% for all the variables.
Chapter 4 Results and Discussions

4.1 Experimental Full-scale Teeming Process

Figure 3 shows the teemed steel weight as a function of the teeming time and the effect of grid refinement on the calculated total teemed steel weight. In order to get grid independent result, grid No. 1 with 25 layers cell in nozzle and grid No. 2 with 15 layers cell in nozzle were calculated by using the realizable $k-\varepsilon$ model with an EWT model. This requires that the first layer cell is located within $y^+<4$ or 5, where $y^+$ is the non-dimensional distance to the wall, defined as $\rho u_t y/\mu$, where $u_t$ is the friction velocity. Grid No. 3 with four layers cell and grid No. 4 with eight layers cell were calculated by the realizable $k-\varepsilon$ model using a SWF. This requires that the first layer cell is located within $12<y^+<60$. The results showed that grid No. 2 with a total number of about 65000 cells and grid No. 4 with a total number of about 30000 cells can be used in combination with the realizable $k-\varepsilon$ model with an EWT model and a SWF model to get grid independent results, respectively.

In Figure 3, data is given for the Bernoulli equation, the inviscid VOF model, the laminar VOF model, the VOF+ realizable $k-\varepsilon$ turbulence model and experimental result\cite{62} to
illustrate the teeming process. Firstly, the results of the Bernoulli equation and mathematical simulations by using the inviscid VOF model were compared. The good agreement between them illustrates the reliability of the mathematical calculations. The pilot-plant experimental results are a little bit smaller than those of the ideal Bernoulli equation and the inviscid VOF model. Furthermore, they are larger than the results from the VOF+ realizable $k$-$\varepsilon$ model with both an EWT model and a SWF model. For the first 200 s, the realizable $k$-$\varepsilon$ model shows a good agreement with the data from the pilot-plant experiment. However, the differences between them increase gradually to around 6% for a case with a SWF model and to 8.5% for a case with an EWT model after around 220 kg steel has been teemed. Considering the experimental condition of steel flowing out from the nozzle outlet exposed in the air, oxidation of steel and steel erosion from the nozzle refractory during the teeming process are inevitable. Therefore, the steel weight obtained from the scale should be a little bit larger than the real steel weight. The Reynolds number range of the experimental full-scale teeming process, beginning with 350 kg steel in ladle and ending with about 130 kg left in ladle, is roughly between 15000 and 20000. Therefore, although the results of the laminar model are closer to the experimental results for longer teeming times, the realizable $k$-$\varepsilon$ turbulence model is believed to be able to better reflect the real flow phenomena in the ladle and nozzle. In addition, the predictions of the realizable $k$-$\varepsilon$ model using a SWF model seems to be closer to the experimental data than the predictions with an EWT model.

**4.2 Steel Flow Velocities Predicted by Different Models with Wall Functions**

The steel velocity distributions in the nozzle are shown in the Figure 4. Predicted data of the velocity distribution is presented for the realizable $k$-$\varepsilon$ model with an EWT, the realizable $k$-$\varepsilon$ model with a SWF and the low Reynolds number $k$-$\varepsilon$ model. Data are given for two cases when 75 kg and 125 kg steel are left in the ladle. From Figure 4b, it can be seen that the realizable $k$-$\varepsilon$ model with a SWF as well as an EWT predicts a larger velocity near the nozzle center when 75 kg steel is left in the ladle than when 125 kg steel is left at the location line 1 in Figure 4a. However, in the near wall region, an opposite result is obtained. This means that when 75 kg steel is left in ladle, much more steel flows
into the nozzle from the center region of the nozzle compared to when 125 kg steel is left in ladle. The low Reynolds number $k$-$\varepsilon$ model does not show similar results. The velocity when 125 kg steel is left in the ladle is larger than that when 75 kg steel is left in the ladle. The predicted maximum velocity at the nozzle center is similar for the two flow models. For the low Reynolds number model, an obvious characteristic exists near the nozzle wall, where a velocity peak is predicted. **Figure 4c** shows the velocity distribution at the location of line 2 in Figure 4a. The realizable $k$-$\varepsilon$ model with a SWF model as well as the EWT model predicts similar velocity distribution patterns, with a slightly different velocity magnitude. The predicted maximum velocity exists near the nozzle wall region rather than at the nozzle center. A possible reason for this phenomenon is the transition of the nozzle geometry, which leads to a significant change of the flow field. A different result is obtained by the low Reynolds number model, but a velocity peak near nozzle wall is also predicted. **Figure 4d** shows the velocity distribution at the location of line 3 in Figure 4a. Similar velocity distributions are predicted by the three different turbulence models in the fully developed turbulent region. However, an obvious difference exists at the near-wall viscosity-affected region. The data illustrates that the realizable $k$-$\varepsilon$ model with a SWF may not make a good prediction of the near wall viscosity-affected flow compared to the EWT model and low Reynolds turbulent model. **Figure 4e** shows the velocity at the nozzle outlet. It seems that a fully-developed turbulent flow is obtained when steel flows out from the nozzle outlet. At the near wall region, the realizable $k$-$\varepsilon$ model with an EWT model and a low Reynolds number model predict a similar steel flow. However, near the nozzle center, the realizable $k$-$\varepsilon$ model with an EWT model as well as the SWF model predicts similar results. **Figure 4f** shows that almost the same steel flow is predicted by realizable $k$-$\varepsilon$ model with the EWT and SWF models at the nozzle center. The velocity magnitude predicted by the low Reynolds number model is smaller than for the other two models at the nozzle center near the outlet.
Figure 4. Steel velocity magnitude distribution in nozzle predicted by the $k$-$ԑ$ model with a standard wall function, the $k$-$ԑ$ model with an enhanced wall treatment and the low Reynolds $k$-$ԑ$ model, (a) ladle geometry, (b) line 1, (c) line 2, (d) line 3, (e) line 4, (f) line 5. SWF and EWT represent standard wall function and enhanced wall treatment, respectively.
Overall, it can be seen from Figure 4 that the geometry transition has a significant influence on the flow field. Therefore, a model coupling the ladle and nozzle or the tundish and nozzle should be an accurate way to reveal the flow reality in the nozzle. The EWT model and the low Reynolds model should predict the steel flow in viscosity-affected near nozzle wall region better, where a transition from a laminar flow to a fully developed turbulent flow occurs. This region is very important for the transport of inclusions. Therefore, an EWT model and a low Reynolds number model should be more proper to predict the inclusion behavior near the nozzle wall due to they can resolve the viscosity-affected boundary layer. However, these two models, in general, require a larger calculation task than the SWF model, due to that finer grids are required at the near wall region. The model with a SWF seems not to predict the flow near the wall very well, but it requires less calculation time. In the remaining part of paper, the realizable $k-\varepsilon$ model with an EWT was used to study the drain sink.

### 4.3 Drain Sink Flow Phenomena at Late Stage of Teeming

At the late stage of a ladle teeming process, a drain sink flow phenomena of steel may result in slag entrainment. Although much effort has focused on this problem\[1, 3, 7, 10, 12, 16, 17]\, there is still no effective way to completely prevent the occurrence of a drain sink flow. However, methods that can reduce the residual steel before the drain sink phenomena occur should be favorable for obtaining a bigger steel yield. A drain sink flow is a kind of flow phenomena, so changing the geometry of the ladle bottom and nozzle should be a possible way to change the flow phenomena at the late teeming stage. This may be an effective way to increase steel yield. In this paper, the drain sink flow phenomena were studied by using mathematical modeling. The VOF model was used to track the interface of steel and gas. The interfaces at different teeming times are shown in Figure 5. It can be seen that a drain sink flow phenomena cannot be observed even when only 0.64 kg steel was left. According to the previous research, the reason for the drain sink is that the volume flow of the open channel at the ladle bottom becomes less than the corresponding outlet capacity.\[7\] The volume flow towards the outlet and the outlet capacity are given by equations (19) and (20), respectively.\[3\]

\[
V = \pi \cdot d \cdot h \cdot v \quad (19)
\]
\[ \dot{V} = \alpha \cdot \pi \cdot \frac{d^2}{4} \cdot \sqrt{\frac{2g \cdot (h + l)}{1 + \beta \frac{l}{d}}} \]  

(20)

where \( d \) is nozzle outlet diameter, \( h \) is steel height in ladle, \( l \) is nozzle length, \( \alpha \) is contraction number, \( \beta \) is coefficient of friction. When the volume flow towards the outlet is smaller than the nozzle outlet capacity, a drain sink flow occurs. For a specific nozzle geometry, the diameter, \( d \), is constant. In order to maintain a big volume flow towards outlet, an increased steel flow velocity is necessary. For the present simulations with a gradually expanding nozzle, the outlet capacity is controlled by the steel height and the nozzle outlet, which has a small nozzle diameter of 5 mm. The steel in the gradually expanding shape nozzle is influenced by the gravity in the steel flow direction. Therefore, steel flow towards the outlet, which satisfies the outlet capacity. Meanwhile, according to equation (20), the outlet capacity also gradually decreases owing to the quick decrease of steel height in the nozzle. Therefore, the gradually expanding nozzle is favorable for obtaining a big steel yield, which is not reported in previous research. However, there are some steel left in the flat ladle bottom during teeming. This steel has a poor ability to flow into the nozzle because of the steel viscosity and the lack of a high enough driving force. In Figure 5(c)-5(d), it is obvious that the residual steel at flat ladle bottom flows slowly into the nozzle. This leads to steel surface fluctuations and air entrainments, which also makes it difficult to check whether the drain sink phenomena happen or not. Methods like an inclined ladle bottom should be an effective way to resolve this problem. This is due to that the gravity in the steel flow direction at an inclined ladle bottom increases the steel flow ability and decreases the residual steel.

In order to verify the above explanation, the teeming process with the commonly used uniform nozzle diameter was simulated. The result is shown in Figure 6. There was about 4.50 kg steel left in ladle when a drain sink happened. At this point, the residual steel height in the ladle was roughly equal to the nozzle diameter, which is in agreement with data reported by other authors\(^3,7,12\)). Most of the previous researches investigated the drain sink flow at ladle with a flat bottom. However, Mazzaferro et al.\(^12\)) researched the influence of an inclined ladle bottom on the drain sink flow by using mathematical modeling. The steel left in ladle is strongly reduced by inclining the ladle bottom.
**Figure 5-7.** Predicted interface at different teeming time by the enhanced wall treatment model.

**Figure 5.** Predicted interface at different teeming time by VOF+ $k$-$\varepsilon$ model with enhanced wall treatment.

<table>
<thead>
<tr>
<th>(a) 7.50 kg steel left</th>
<th>(b) 1.80 kg steel left</th>
</tr>
</thead>
<tbody>
<tr>
<td>(c) 1.05 kg steel left</td>
<td>(d) 0.64 kg steel left</td>
</tr>
</tbody>
</table>

**Figure 6.** Predicted interface at the beginning of drain sink with uniform diameter nozzle.

- About 4.50 kg steel left

**Figure 7.** Predicted interface at different teeming time in an inclined bottom ladle by VOF+ $k$-$\varepsilon$ model with enhanced wall treatment.

<table>
<thead>
<tr>
<th>(a) 5 kg steel left</th>
<th>(b) 0.62 kg steel left</th>
</tr>
</thead>
<tbody>
<tr>
<td>(c) 0.20 kg steel left</td>
<td>(d) 0.025 kg steel left</td>
</tr>
<tr>
<td>(e) 0.01 kg steel left</td>
<td>(f) 0.004 kg steel left</td>
</tr>
</tbody>
</table>

Note: All the pictures have the same color map as Figure 5(a) and only the pictures of ladle bottom and nozzle were cut to show.
Table 4. Enlarged pictures of the oval circle in Figure 6-7.

<table>
<thead>
<tr>
<th>Figure 6</th>
<th>Figure 7(d)</th>
<th>Figure 7(e)</th>
<th>Figure 7(f)</th>
</tr>
</thead>
</table>

However, the geometry of nozzle is not considered in their research. The teeming process in a 2 degree inclined bottom ladle with a gradually expanding nozzle was simulated in this paper and shown in Figure 7. It can be seen that no drain sink flow happened during the whole process. Even though the steel surface is inclined, there is no gas entrained into steel. From the comparison of Figure 5 and Figure 7, it is obvious that a great amount of steel left in the flat ladle bottom with 1.05 kg steel in Figure 5(c) and 0.20 kg steel in Figure 7(c) for the same steel surface height in nozzle. Therefore, the combination of an inclined ladle bottom and a gradually expanding diameter nozzle should be an effective way to improve the steel flow ability, which is needed for preventing a drain sink flow to develop and to obtain a bigger steel yield. However, it is not clear how freezing slag and steel at ladle bottom after teeming could affect an inclined ladle bottom during operation. Furthermore, the 3-D simulation should be done in the future to further ascertain the 2-D simulation results.

4.4 Turbulence Properties of Steel Flow during a Teeming Stage

The steel flow field was firstly solved. The properties of the steel flow field in the nozzle at a point when around 300 kg steel is left in the ladle are shown in Figure 8-11. Figure 8 and Figure 9 show the turbulent kinetic energy and turbulent dissipation rate of the steel flow field in the nozzle, respectively. It can be seen that the turbulent properties reach their maximum values approximately at the connection region of the straight pipe part and the expanding part of the nozzle (region 5). This illustrates that the steel flow is very chaotic in this region. The biggest shear stress value also exists around this region, as is shown in Figure 10. More specifically, it illustrates that the steel flow has a large stress force on the nozzle wall. As reported by Uemura et al.\textsuperscript{63}, an inclusion that comes close to a nozzle wall sinters and forms a neck with the refractory of a nozzle wall or with an inclusion that already deposited on a nozzle wall. According to Sambasivam\textsuperscript{42}, the
neck will be broken and the inclusion may be washed away when a shear stress at the neck is higher than the sinter bond strength. Therefore, in the regions with a high shear stress level, a deposition of an inclusion may be prevented even though the inclusion is present or moving close to the nozzle wall.

**Figure 8-11.** Predicted properties of the steel flow field at a time when around 300kg steel is left in ladle.

**Figure 8.** Turbulent kinetic energy.

**Figure 9.** Turbulent dissipation rate.

**Figure 10.** Shear stress.

**Figure 11.** Velocity contour.

Note: Figure (b) in the Figure 8-11 is the enlarged pictures of the oval enclosed parts of the corresponding Figure (a).

**Figure 11** shows the velocity distribution of the steel flow in the nozzle. The quick change of velocity contour in around the region 5 shows a high velocity gradient, which means that a turbulent flow developed very quickly. In the connection regions, where a
nozzle geometry transition exists, a sudden change of the steel flow direction occurs. The velocity of radical steel flow greatly decreases due to its collision with a downwards directed steel flow at the nozzle core part. This kind of collision also increases the flow chaos, contributing to the turbulence level of the steel flow.

4.5 Inclusions Tracking Neglecting a Stochastic Turbulent Motion of Inclusions

Six different sizes of inclusions, 0.5 µm, 3 µm, 10 µm, 20 µm, 100 µm and 400 µm, released from location 3 in Table 2 were tracked using a Lagranian method under the previously obtained fixed flow field. As mentioned before, it is reasonable to use the fixed flow field due to that only a small change of the flow field occurs during the short time that the inclusions pass through the nozzle. In addition, the focus is to compare the inclusion behaviors in the same flow field. In order to obtain a clear view on the flow abilities of different sizes of inclusions, a stochastic turbulent model for inclusion movement is not used at first. In this way, the uncertainty that a stochastic turbulent random motion leads to is avoided.

Figure 12. Locations of inclusions at different times, (Coordinate origin is located at the center of the ladle).
Figure 12 shows the locations of inclusions at different times in the nozzle. It can be seen that inclusions with a diameter 0.5 µm, 3 µm, 10 µm, 20 µm and 100 µm have similar trajectories. However, for an inclusion with a 400 µm diameter, the trajectory is obviously different from other inclusions. It moves closer to the nozzle center and takes a much longer time before it reaches the nozzle region, as can be seen in Figure 13(b), than the other inclusions. The behaviors of inclusions are mainly determined by three forces: i) an inertia force, ii) a drag force and iii) a buoyancy force due to a density difference between an inclusion and steel. In the current situation, the angle between the upwards buoyancy force and the downwards drag force is larger than 90°. Therefore, the drag force in the z direction needs to combat the buoyancy force to make inclusions move to the nozzle region. A larger-size inclusion has a bigger buoyancy force than a smaller-size inclusion. Meanwhile, the steel flow velocity is small at the release location 3, which leads to a smaller drag force due to a smaller magnitude of velocity difference between steel and inclusion. The relative magnitude of these two forces will determine the inclusion behavior. In order to explain the obviously different behaviors of 400 µm inclusions compared to other sizes of inclusions, the buoyancy force and the drag force for 100 µm and 400 µm inclusions at the release location 3 as well as at the straight pipe location were calculated, as is shown in Table 5. At the release location 3, it can be seen that the downwards drag force of 400 µm inclusions has a similar magnitude to the upwards directed buoyancy force. However, a much larger drag force than a buoyancy force was obtained for the 100 µm inclusions. This means that 100 µm inclusions can move much faster in the downwards z direction compared to 400 µm inclusions. The competition of these two forces in the z direction makes 400 µm inclusions to take a longer time than smaller inclusions, like 100 µm inclusions, to move to the nozzle region. This also gives them more time to move towards the nozzle center, as is shown in Figure 12. The larger inertia of big inclusions than that of small inclusions also causes them to take a longer time to respond under the same conditions. At the straight pipe location, around 0.026 m from the nozzle outlet, the drag forces of both 100 µm and 400 µm inclusions are much larger than the buoyancy forces, which causes them to move fast in the nozzle pipe region.
Table 5. Buoyancy force and drag force of inclusions in the z direction.

<table>
<thead>
<tr>
<th>Location</th>
<th>Size, µm</th>
<th>Buoyancy force, N</th>
<th>Drag force, N</th>
<th>Acceleration a, m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>Release location 3</td>
<td>100</td>
<td>-1.02×10^{-9}</td>
<td>1.50×10^{-8}</td>
<td>3.93</td>
</tr>
<tr>
<td></td>
<td>400</td>
<td>-6.53×10^{-8}</td>
<td>6.70×10^{-8}</td>
<td>7.66×10^{-3}</td>
</tr>
<tr>
<td>Pipe location, 0.026m from nozzle outlet</td>
<td>100</td>
<td>-1.02×10^{-9}</td>
<td>1.50×10^{-7}</td>
<td>36.79</td>
</tr>
<tr>
<td></td>
<td>400</td>
<td>-6.53×10^{-8}</td>
<td>2.09×10^{-5}</td>
<td>91.70</td>
</tr>
</tbody>
</table>

The change of inclusion velocities in the y direction, parallel to the cross section of the nozzle, and the z direction, vertical to the cross section of the nozzle, as a function of time are shown in Figure 13(a) and (b), respectively. The characteristics of the inclusion velocities in the x direction, which is not shown in this dissertation, are similar as those in the y direction, except with respect to the velocity magnitude. It can be seen that inclusions with a diameter of 0.5 µm, 3 µm, 10 µm and 20 µm have a similar velocity pattern with a minor difference of magnitude at the same elapsed time, which means that they have similar trajectories, which was previously shown in Figure 12. With the increase of inclusion sizes, especially for inclusions larger than 100 µm, the maximum velocities of inclusions in both the y and z directions decrease. This can clearly be seen in Figure 13(c) and (d). In the y direction, the main reason for that is the inertia of inclusions. The smaller fluctuation of the y-direction velocity of big inclusions in Figure 13(c) illustrates the larger influence of inclusion inertia on their motion. For the z-direction inclusion velocity, both the inertia force and the buoyancy force should be responsible for a little bit smaller velocity magnitude for the larger inclusions than for the smaller inclusions. From Figure 13(c), it can be seen that a sharp decrease of the y-direction inclusion velocities occurs at the locations of 3 cm and 7 cm away from the nozzle outlet, where the connection regions of the nozzle exist. As previously mentioned, a rapid decrease of steel velocity in the y direction should be the reason for that. In Figure 13(d), the inclusion velocity increases rapidly within a 2 cm distance, going from 5 cm down to 3 cm distance from the nozzle outlet. This location is situated just above the straight pipe part of the nozzle. This also illustrates that the turbulence intensity increases quickly in this region.
Figure 13. Velocities of inclusions in nozzle, (a) and (b) are y-direction and z-direction velocities of inclusions as a function of time, respectively; (c) and (d) are y-direction and z-direction velocity distributions of inclusions at different distance from the release location 3, respectively.
4.6 Inclusion Behavior Including Stochastic Turbulent Motions

The behaviors of inclusions in a nozzle were statistically analyzed. In order to understand the inclusion behaviors at different nozzle regions, statistical analysis was also carried out to investigate the sensitivity of different nozzle regions on the possible inclusions deposition. Considering the result of 4.5 and a small number of large-size inclusions, e.g. bigger than 100 µm, existing in steel, three sizes of inclusions, 1 µm, 10 µm and 100 µm, were tracked. Twenty inclusions were released from each location in Table 2.

![Figure 14](image)

**Figure 14.** Number of inclusions that touch the nozzle wall, (a) influence of all 5 release locations on the number of inclusions touching the nozzle wall, (b) number of different-size inclusions from all release locations touching the nozzle wall, (c) and (d) influence of inclusion sizes from all release locations and release locations for all sizes of inclusions on the number of inclusions touching the nozzle wall at each nozzle region.
The number of inclusions that touch the nozzle wall is shown in Figure 14. Figure 14(a) shows the influence of inclusion release locations on the number of inclusions touching the wall. It can be seen that some inclusions released from location 1 and 2 touch the nozzle wall. However, there is no inclusion, released from location 3, 4 and 5, touching the nozzle. This illustrates that the deposition possibility of inclusions from location 3, 4 and 5 is very low. The inclusions from location 1 have the highest possibility of touching the nozzle wall among all the release locations. One possible reason is that location 1 is close to the nozzle wall, which provides an inclusion with a short distance to pass through to the nozzle wall. Figure 14(b) shows the number of different-size inclusions from all the release locations touching the nozzle wall. For each size of inclusions, 10 µm inclusions seem to have a little bit higher possibility to touch the nozzle wall than the other two sizes of inclusions. Figure 14(c) and 14(d) show the influence of inclusion sizes and release locations on the number of inclusions touching the wall of each nozzle region. It can be seen that for the 1 µm inclusions, the distribution of inclusions that touch the nozzle wall along the nozzle height is more uniform than the other two sizes of inclusions. For 10 µm and 100 µm inclusions, two nozzle regions, region 1 and region 5, have a higher number of inclusions touching the nozzle wall compared to the other regions. This illustrates that region 1 and region 5 have a higher possibility for nozzle clogging than the other regions. In region 1, the steel flow velocity is very small. The turbulence properties in Figure 8 and 9 also show that turbulence intensity is not high in this region. Furthermore, Figure 14(d) shows that all the inclusions that touch the nozzle wall within regions 1 to 4 are from release location 1. Therefore, the following reasons should be responsible for a large number of inclusions moving to the wall in nozzle region 1: 1) the release location 1 is close to the wall of nozzle region 1; 2) turbulence plays a positive role for the transport of inclusions to the wall; 3) centripetal force, which has also been investigated by Wilson[26], should be helpful for the inclusion transport during flow direction changing in region 1. In region 5, as is previously shown in Figure 11, steel velocity changes quickly. Therefore, the steel flow turbulence developed very fast. Furthermore, the collision between the radial steel flow and the downwards core flow contributes to the flow turbulence. Inclusions are transported to the nozzle wall by turbulent eddies. Even the radial directed steel flow moves inclusions directly close to the
nozzle wall in region 5. A smaller diameter of the nozzle in this region gives inclusions a shorter distance to pass through to reach the nozzle wall. Both the high turbulence intensity and the short moving distance cause inclusions easy to touch the nozzle wall. For the inclusions released from location 2, it can be seen from Figure 14(d) that inclusions only touch the nozzle wall in region 5 and region 6. As previously mentioned, a high turbulence as well as a short moving distance should be the reason for that. However, a higher steel flow velocity in region 6 than that in region 5 causes inclusions to have a shorter residence time in the straight pipe nozzle part. This short residence time causes some inclusions not to have enough time to move to the nozzle wall in region 6, even though the flow turbulence is also high in region 6.

![Figure 15](image1)

**Figure 15.** Pictures of a part of the clogged nozzle that correspond to region 5 and 6 in Figure 2(b). ((a) and (b): Erik Roos, personal communication, January 10, 2013)

The experimental results, as is shown in Figure 15, illustrate the reliability of this simulation work and the above explanation. It can be seen that a serious clogging is found in region 5, which is in a good agreement with the present simulation results. Region 1 is located at the connection part of the nozzle and ladle bottom. After experiment, the steel left in ladle must be tapped out making it difficult to get a sample of solidified steel at that region. However, supporting the model results is the comparable experiments
reported by Kojola[52], demonstrating that clogging also frequently occurs in the upper part of the nozzle. Both the simulation and experiment show that the transition region of geometry, or flow field, is the sensitive region for an inclusion deposition, as well as for clogging.

The statistical analysis in this study gives some information on inclusion behaviors in nozzles. Even though it is not clear from the results of this study on whether inclusions deposited or not after touching the nozzle wall, it provides some information on where inclusions touch the nozzle wall and what places should be sensitive for nozzle clogging. Thus, a future deposition model should be developed to describe the inclusion behaviors when they come close enough to the nozzle wall. Also, when more complex production systems are to be studied using particle (inclusion) tracking, a stochastic model that is exactly compatible with properties of the crossing-trajectory effect as determined by Kolmogorov similarity scaling for high Reynolds-number turbulence could be used. As was pointed out by Reynolds et al.[64], model predictions for particle dispersion in grid generated turbulence are shown to be in close agreement with experimental data of Snyder and Lumley[65] and integral timescales are shown to be compatible with the much used parameterizations advocated by Csandy[66] and by Frenkel[67]. In the future, perhaps the most suitable model for the evolution of fluid velocities along particle trajectories in complex metallurgical production systems would be the Langevin equation[68] used in conjunction with velocity fields calculated using the Eulerian approach.
Chapter 5 Conclusions

In this thesis, steel flow phenomena and inclusion behavior during a ladle teeming process were studied by using mathematical simulation and experiment. The focus of this work is on the steel flow and inclusion motion in nozzle. Firstly, the teeming process was simulated and compared by using different turbulence models with different wall functions. Steel flow velocities predicted by different models were also compared. A proper verified model by experiment was used to study the drain sink flow phenomena. Then, the turbulence properties of steel flow in nozzle at a teeming stage were predicted. Inclusion behavior in nozzle was studied by using a Lagranian method under a fixed flow field at the teeming stage. Behaviors of different-size inclusions released from different nozzle locations at different nozzle regions were investigated. The conclusions are summarized as follows:

- Three models were used in the simulations: i) the EWT realizable $k-\epsilon$ model, ii) the SWF realizable $k-\epsilon$ model, and iii) the low Reynolds number $k-\epsilon$ model. The results showed that all three models can simulate the teeming process with a good agreement with the experimental data.
- The realizable $k-\epsilon$ model with an EWT and the low Reynolds number model should be good for the prediction of steel flow at the viscosity-affected region near the nozzle wall. Steel flow in this region is quite important for the inclusion deposition.
- An inclined ladle bottom with a gradually expanding diameter nozzle was theoretically effective to alleviate a drain sink flow, because this kind of geometry increased the steel flow ability towards nozzle outlet because of a large gravity in the flow direction.
- Turbulent properties reach their maximum value in a region around the connection region of the straight pipe part and the expanding part of the nozzle. A high velocity gradient exists in this region, which illustrates that the turbulence develops very fast. At the connection part, the radial directed flow velocity rapidly decreases due to flow collision with the downwards directed steel flow, which increases the flow turbulence.
- It was found that 0.5μm, 3μm, 10μm and 20μm inclusions have similar trajectories and velocity distributions in the nozzle. However, the trajectories of larger inclusions (400μm) were quite different from the smaller ones. Both the inertia force and the buoyancy force play a very important role for the behaviors of large-size inclusions.

- The result of the statistical analysis performed for 1μm, 10μm and 100μm inclusions shows that inclusions that enter the nozzle inlet from a close-wall location have a high probability of touching the nozzle wall. The nozzle inlet region and the connection region of the straight pipe and the expanding part of the nozzle were found to be the sensitive regions for the inclusion deposition as well as for nozzle clogging.
**Future Work**

In order to have a good understanding on the inclusion behavior in steel flow, the following work needs to be carried out in future.

- Water model experiment should be carried out to measure velocities and turbulent characteristics and to compare with the turbulence models. A proper turbulence model, boundary condition and turbulence fluctuation should be considered for the prediction of inclusion behavior.

- A proper deposition model should be developed to describe the inclusion behaviors when they come close to the nozzle wall.

- A proper inclusion tracking method should be considered. The suitable model for the evolution of fluid velocities along particle trajectories in complex metallurgical production systems may be the Langevin equation used in conjunction with velocity fields calculated using the Eulerian approach.
References

SUPPLEMENT 1

Simulations of the Ladle Teeming Process and Verification with Pilot Experiment

Peiyuan Ni, Lage T.I. Jonsson, and Pär G. Jönsson

Accepted for publication in Steel Research International.
Turbulent Flow Phenomena and Ce\textsubscript{2}O\textsubscript{3} Behavior during a Steel Teeming Process

Peiyuan NI, Lage T.I. JONSSON, Mikael ERSSON and Pär G. JÖNSSON

Accepted for publication in ISIJ International.