On the Study of a Liquid Steel Sampling Process

Zhi Zhang

Doctoral Dissertation
Stockholm 2010

Division of Applied Process Metallurgy
Department of Materials Science and Engineering
Royal Institute of Technology
SE-100 44 Stockholm
Sweden
Zhi Zhang  *On the Study of a Liquid Steel Sampling 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--10/34--SE+APRME/TUAVH  

© The Author
Abstract

The liquid steel sampling method is one of the commonly used procedures in monitoring the steelmaking process. Besides it can be used for analyzing the dissolved alloys, hydrogen content and oxygen content, it can be also employed to monitor the inclusion characteristics at the steelmakings. Here, a crucial point is that the steel sampler should be filled and the metal solidifies without changing the inclusion characteristics. Therefore, the objective of this work is to fundamentally understand the liquid steel sampling process by means of analyzing and modeling the two-phase flow during the sampler filling process, and verifying the mathematical model by using the experimental data.

The present dissertation presents an experimental and theoretical study of the filling process of both the lollipop-shaped sampler and the rectangular-shaped sampler. Firstly, a physical modeling by using a water model has been carried out to fundamentally investigate the flow pattern inside the sampler vessels during its filling. The flow patterns were obtained by a PIV system. Then, a mathematical model has been built to theoretically understand the phenomena. The commercial CFD code was used. Here, different turbulence model have been compared between the realizable $k$-$\varepsilon$ turbulence model and Wilcox $k$-$\omega$ turbulence model. It concludes that the Wilcox $k$-$\omega$ turbulence model agrees well with the PIV measurements. Thus, the preferred it was further employed to predict the turbulent flow inside the production lollipop-shaped sampler fillings. It is important to find that the average collision volume in the production steel sampler without solidification at filling is about 30 times higher than that in a ladle furnace.

In the end, the whole sampling system was modeled. The initial solidification during the filling was taken into account. Focus was on the influence of the initial solidification on the inclusion concentrations. A discrete phase model was used to simulate the movement of inclusions in the liquid steel. Some selected different sized primary inclusions that exist in the ladles at a steelmaking process were simulated.

The same method of studying the filling procedure of the lollipop-shaped sampler was further applied to comprehensively investigate the rectangular-shaped sampler.

Key Words: liquid steel sampler, liquid steel sampling process, flow pattern, PIV, physical modeling, mathematical modeling, CFD, solidification, turbulence, Wilcox $k$-$\omega$ model, inclusion concentrations
First of all, I would like to express my sincerely gratitude and appreciation to my supervisor, Prof. Pär Jönsson. Your great patience, professional guidance and constant assistant are strong supports for me during my life at KTH. You really taught me how to conduct research.

I am very appreciated all the assistants from my tutor, Dr. Anders Tillander. Your scientific discussions are very helpful for all my works. I would like to thank Dr. Mikael Ersson for the assistant of simulations and discussion of computer codes. Thank you for arranging the most popular Swedish sports—“innebandy”, which makes the study at KTH with lots of expectations.

I also want to express my gratitude to Prof. Manabu Iguchi at Hokkaido University for providing me an opportunity to carry out my research in Japan.

Many thanks to Dr. Andrey Karasev for the very nice discussion concerning on the inclusions that makes me feel easy to deal with the large amount of data. Thanks to Prof. Hasse Fredriksson for the discussions and suggestions on the simulation of solidifications.

Special thanks to Prof. Shinichiro Yokoya for the help on my work at the beginning of my Ph.D study. I also appreciate the help from Prof. Lage Jonsson.

I have to thank my roommate in the division, Mr. Ola Ericsson, for providing me the very important data of plant trails as well as the useful discussion on our project. Thanks to all the other colleagues for sharing such a pleasure time in the division of TPM.

Finical support by the European Union under the contract number RFSR-CT-2007-0005 and Research Fund for Coal and Steel of the European Community are acknowledged.

Last but not least, many thanks to my parents in China, for their never-ending love, support and encouragement.

Zhi Zhang

Stockholm, August 2010
SUPPLEMENTS

The present dissertation is based on the following papers:

**Supplement 1:** Zhi Zhang, Anders Tillander, Manabu Iguchi and Pär Jönsson, “Physical Modeling of a Sampler Filling for Molten Steel”, *ISIJ Int.*, **49** (2009), No. 10, 1522

**Supplement 2:** Z. Zhang, A. Tilliander and P.G. Jönsson, “Mathematical Modeling of Water Sampler Filling”, *Steel Research Int.*, **81** (2010), No. 2, 112

**Supplement 3:** Z. Zhang, A. Tilliander and P. G. Jönsson, “Simulation of the Filling of a Liquid Steel Sampler”, *Steel Research Int.*, **81** (2010), No. 9, 749

**Supplement 4:** Zhi Zhang, Anders Tilliander, Andrey Karasev and Pär G. Jönsson, “Simulation of the Steel Sampling Process”, *in press, ISIJ Int.*, **50** (2010), No. 12

**Supplement 5:** Zhi Zhang, Anders Tilliander, Manabu Iguchi and Pär G. Jönsson, “An Experimental and Numerical Study of the Filling of A Steel Sampler”, manuscript, *ISRN KTH/MSE--10/35--SE+APRMETU/ART*

The contributions by the author to the different supplements of the dissertation:

1. Literature survey, experimental work, major part of the writing;
2. Literature survey, numerical calculations, major part of the writing;
3. Literature survey, numerical calculations, major part of the writing;
4. Literature survey, numerical calculations, major part of the writing;
5. Literature survey, experimental work, numerical calculations major part of the writing
Parts of the work have been presented at the following conference:

   “Simulation of a Sampler Filling”
   **Z. Zhang**, S. Yokoya, A. Tilliander and P. Jönsson

2. Third International Conference on Simulation and Modeling of Metallurgical Processes,
   Sep. 8-9, 2009, Leoben, Austria
   “An Experimental and Theoretical Study of the Filling of a Steel Sampler”
   **Z. Zhang**, A. Tilliander, M. Iguchi and P. G. Jönsson

3. 2nd International Symposium on Cutting Edge of Computer Simulation of Solidification
    and Casting, Feb. 3-5, 2010, Sapporo, Japan
   “Simulation of the Steel Sampling Process”
   **Zhi Zhang**, Anders Tilliander, Andrey Karasev and Pär G. Jönsson

List of papers not included in this dissertation:

1. L. Hallgren, **Z. Zhi**, S. Yokoya, S. Takagi, A. Tilliander and P. Jönsson, “Use of Swirl
   Phenomena in Ingot Casting to Improve Filling Conditions”, submitted to *ISIJ International* to publish, 2010

   Blade Effects in Uphill Teeming Casting”, *in press, ISIJ Int.*, **50** (2010), No. 12
## CONTENTS

*Chapter 1 INTRODUCTION* ................................................................. 1
  1.1 Principle of Liquid Steel Sampling Procedure .................................. 2
  1.2 Objective and Overview of the Work ............................................. 4

*Chapter 2 PHYSICAL MODELING* ...................................................... 7
  2.1 Experimental Setup ...................................................................... 7
  2.2 Measurements of Flow Fields ...................................................... 9

*Chapter 3 MATHEMATICAL MODELING* ............................................. 11
  3.1 Numerical Assumptions .............................................................. 11
  3.2 Numerical Methods ..................................................................... 11
  3.3 Boundary Conditions .................................................................. 13
  3.4 Properties of Materials ............................................................... 14
  3.5 Methods of Solutions .................................................................. 14

*Chapter 4 RESULTS* .......................................................................... 15
  4.1 Experimental Results of Physical Modeling ..................................... 15
  4.2 Water Flow Simulation and Mathematical Validation ....................... 19
  4.3 Simulation of Steel Flow in the Lollipop-Shaped Sampler without Solidification ...... 23
  4.4 Simulation of Steel Flow with Initial Solidification in the Lollipop-Shaped Mold ...... 27
  4.5 Dispersions of Inclusions ............................................................. 29

*Chapter 5 DISCUSSION* ..................................................................... 35

*Chapter 6 SUMMARY & CONCLUSIONS* ........................................... 41

*FUTURE WORK SUGGESTIONS* ......................................................... 43

*Nomenclature* ................................................................................... 44

*Reference* ......................................................................................... 45

*Appendix* ......................................................................................... 46
Chapter 1

INTRODUCTION

In recent years, much effort has been made to investigate the inclusion characteristics during steelmakings to produce clean steels. It is now realized that an online control of the inclusion size distributions during steelmaking is significantly crucial for the final property control of steel. Therefore, it is important for the steelmakers to indentify various types of inclusions present in the melt in a very early stage during the steelmaking process, especially during the secondary steelmaking. Here, it should be noted that non-metallic inclusions are inevitably present in the liquid steel. However, in order to improve the material properties of the final product, they should be removed to the largest possible degree before casting for the majority of steel grades. The most commonly used technique to assess the inclusion size distribution is the liquid steel sampling procedure followed by a microscopic analysis.

The liquid steel sampling is one of the most important procedures in order to monitor the composition of the online steelmaking, oxygen and hydrogen contents. Feedbacks from an analyzed steel sample are immediately sent to the steelmakers to perform online controls. It is possible to achieve sampling, analyzing and receiving feedbacks on the steel composition within a short time (usually less than 5 min). Characteristics of all the elements dissolved in the steel can be determined throughout the process. Similarly, it is also possible to measure the temperature accurately throughout the entire metallurgy process. However, there is no practical method available currently for determining and monitoring inclusion characteristics throughout the steelmaking process as a part of the process control. This is due to that it is very time consuming to access the total oxygen content and size distributions of inclusions. Table 1-1 shows the opportunities to obtain metallurgical feedback during the steelmaking process. It is clear that no information or methods available to obtain the information of inclusion characteristics online.

<table>
<thead>
<tr>
<th>Table 1-1 Opportunities for metallurgical feedback during steelmaking</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ladle</td>
</tr>
<tr>
<td>-----------------</td>
</tr>
<tr>
<td>Elements dissolved in the steel</td>
</tr>
<tr>
<td>Temperature</td>
</tr>
<tr>
<td>Inclusion characteristics</td>
</tr>
</tbody>
</table>

Therefore, more investigations have been done to determining not only the composition, but also the inclusion characteristics from the extracted steel samples. Thus, the function of a common steel sampler is extended to include an analysis of the inclusion characteristics. Efforts have been extended in developing novel methods to achieve the aim of rapid analysis.
online by using the same liquid steel sampler mold. One of the latest methods to quickly
detect the inclusions on a sample is the OES-PDA (optical emission spectrometry with pulse
discrimination analysis) method.\textsuperscript{2} It uses spark discrimination of the elemental intensities
present as the soluble element and in the inclusions. Within only 7 min, the analyzed results
can be obtained. By applying this method, the steelmakers can accurately control the process.
Thus, much cleaner steel can be obtained which, in turn, can improve the final steel properties
significantly.

1.1 Principle of Liquid Steel Sampling Procedure

1.1.1 Production sampling process in the industry

For the production, a lollipop-shaped sampler mold is used. It is encased in the baked
sand at one end of a sampler rod which penetrates through the slag on the top of a ladle and
immerses in the molten steel as shown in Figure 1-1. Then, the liquid steel will be, pressed by
the ferrostatic pressure or, sometimes when argon gas used, extracted by applying a
backpressure into the steel sampler mold. A schematic plot of a lollipop-shaped steel sample
and its sampling system is illustrated in Figure 1-2. Here, it is important to know that the
filling of a liquid steel sampler is a dynamic process with the following macro-phenomena:

- High temperature
- Fast solidification
- Reoxidation
- Turbulence flow.

The whole process of extracting liquid steel from a ladle, for example, can be roughly
divided into three steps:
1. The sampler is heated to a temperature of approximately 110°C to ensure it is completed dry. Sometimes argon is used to blow away the air inside the sampling rod;

2. Argon gas is flushed through the sampling lance before the immersion of sampler in the melt. Hence, the pressure is set to balance the ferrostatic pressure at the sampling depth and avoid the undesired top slag flowing into the sampler mold. If without applying argon protection, usually there is a lip covering the inlet of the sampler mold to avoid the top slag during the penetration through the slag region on top of the ladle. The lance moves down, penetrating through the slag and keeping its head immersed in the molten steel. Thereafter, it stops shortly after the steel sensor senses the steel melt;

3. As soon as the inlet of mold has reached the position of a pre-determined depth, a pressurized gas is introduced by an ejector connected to the handle of the sampling rod to invert the argon flow. However, due to the baked sand used around the outside of sampler mold (shown in Fig. 1-2), the pressure inside the sampler mold may not decrease dramatically. This is because the argon gas must penetrate into the sand first and then flow out. The sand is similar to a porous media. As a consequence, the filling of the sampler, which is also partly contributed by the ferrostatic pressure, is carried out under controlled conditions. After a holding time of about three to five seconds, the sampler rod will be lifted vertically from the melt;

![Fig. 1-2 Schematic plot of a lollipop-shaped steel sample and part of a sampling system](image)

1.1.2 The sampling process in the laboratory

The study of the filling process of a rectangular-shaped sampler mold has also been carried out. The prepared steel sample is used to study grain refining of steel by using inclusions as inoculants. The present research represents a small part in a larger effort to contribute the study. Here, steel samples are sucked up from a steel melt and then the inclusion characteristics and grain size are determined. The molten steel is prepared in a small scaled furnace in the laboratory. A Φ6 mm sized inlet pin connected to the bottom center of the main rectangular body penetrates into the liquid steel in the furnace. The sampling is
carried out by exhausting the rectangular-shaped sampler through a pipe on top. Thus, the sampler can be filled by the assistant of atmosphere pressure.

However, it is important to keep in mind that the filling conditions must be calm in order to minimize the risks of reoxidation, which in turn causes the formation of new inclusions that do not actually exist in the steel at the time of sampling. Moreover, it should also be mentioned that it is important to study the turbulent flow. The turbulent flow, which is characterized as a fluctuating flow pattern, will lead to more opportunities for the inclusions to collide and grow during fillings. Thus, smaller sized inclusion will grow into bigger ones, and as a result incorrect information from the steel sample might be given. This, in turn, might lead to wrong decisions in the steelmaking, for example, more gas stirring time may be applied.

1.2 Objective and Overview of the Work

As mentioned above, the difficulties of carrying out laboratory scaled experiments to investigate the sampler fillings lie in the complex phenomena taking place during the sampling process. Also, it is difficult to know the inclusion concentrations during the steel making when the steel is in a molten state. The data obtained for the concentrations of the inclusion are by analyzing the solidified steel. Therefore, the most proper approach is by modeling and simulating of this process.

The objective of this work is to fundamentally understand the liquid steel sampling process by means of analyzing and modeling the two-phase flow during the sampler filling process, and comparing the mathematical model with experimental data. The two phases present are air/water or air/liquid steel. In this study, the flow field was considered to be isothermal and incompressible. The initial solidification of steel was also considered. The discrete phase model was employed to simulate the non-metal inclusions. The main features of the work are:

1. Physical modeling of the filling of a lollipop-shaped water sampler vessel has been carried out to fundamentally understand the flow pattern during the filling process of this specific sampler. Different flow fields have been characterized. This is described in detail in Supplement I. Due to the kinematic viscosity similarity between the water at room temperature and the molten steel, water was used to simulate the liquid steel behavior during the experiments. Employing water as a media to carry out experiments in the iron & steel industry has been widely applied in resent years.

2. A mathematical model has been developed to describe the phenomena of the physical modeling which is discussed carefully in Supplement II. The computational fluid dynamics (CFD) method was employed to investigate the flow pattern. Here, it is should be noted that recently, extensive work have been carried out by applying CFD to examine the flow pattern during the metallurgy process at the steelmaking. For example, it has been found that the realizable $k$-$\varepsilon$ model for the turbulence prediction is appropriate for simulations of water flow inside experimental water
vessels. Furthermore, it is also proper for simulations of a liquid steel flow inside the vessels used in productions such as the ladle, AOD converters, the tundish and the continuous casting mold.\(^4, 6-10\) However, there are several other models to describe the flow pattern other than the \(k-\varepsilon\) turbulence model. Therefore, the main purpose of Supplement II is to investigate which turbulence model best predicts and agrees the flow fields obtained by the results of physical modeling.

3. **Supplement III** presents a numerical calculation of the filling process of the lollipop-shaped production sampler using liquid steel. The calculation has been carried out by using the result from Supplement II and assuming the constant temperature of molten steel without solidification and heat transfer. The main aim is to study the flow pattern, especially the turbulent flow. The collision volume between particles due to the turbulence was calculated. Moreover, highly possible particle collision areas were determined and investigated.

4. **Supplement IV** is an extension study of the work done in Supplement III, where the heat transfer and solidification were considered. Moreover, the inclusion dispersions were also taken into account in the calculation. Different areas of top, middle and bottom at surface and center regions of sample were examined. It concludes the best regions in the steel sample for determining the inclusion characteristics;

5. The liquid steel filling process of a rectangular-shaped steel sampler which is used to investigate the grain refining of steel has been studied physically and numerically. The rectangular-shaped steel sampler is designed for use in the laboratory only. The same Wilcox \(k-\omega\) turbulence model was applied. The study procedure was the same as the routine for studying the filling process of lollipop-shaped sampler. All the turbulence, heat transfer, solidification and inclusion dispersions were considered and the results have been presented in Supplement V.

**Figure 1-3** shows a flow chart of the present work and the connections of the different supplements.
Fig. 1-3 Flow chart describing the present work
Chapter 2

PHYSICAL MODELING

2.1 Experimental Setup

The dimensions of both the production lollipop-shaped sampler and the laboratory rectangular-shaped sampler were used for the constructions of the water models. A sketch of the lollipop-shaped sampler including important dimensions is shown in Fig. 1-2. A production sampler typically has a 34 mm diameter of the body part ($\Phi$), a 6 mm diameter of the inlet pin ($d_n$) and a 12 mm (W) thickness. The rectangular-shaped sampler used in laboratory has the following dimensions: $24mm \times 24mm \times 240mm$. These are relatively small geometries that are quite difficult to study by using physical modeling. Thus, it is necessary to scale up the size of the water model in order to simplify the experiments.

In these experiments, the different flow rates were calculated according to the Froude number similarity:

$$Fr_{m,r} = Fr_{m,m}$$  \hspace{1cm} (Eq. 2-1)

where $Fr_{m,r}$ is the modified Froude number in a production sampler and $Fr_{m,m}$ is the modified Froude number in the physical model of a sampler. Calculations of the Fr should be considered according to three different regions in a filled vessel as shown in Figure 2-1. Thus, the modified Froude number should be expressed as follows:

- **Region I: Inlet of mold**
  $$Fr_{m,m,1} = \frac{Q_l^2}{g \cdot d_n^3}$$  \hspace{1cm} (Eq. 2-2)

- **Region II: Middle of mold**
  $$Fr_{m,m,2} = \frac{Q_l^2}{g \cdot D_h^4 \cdot H_L}$$  \hspace{1cm} (Eq. 2-3)

- **Region III: Surface of liquid in the mold**
  $$Fr_{m,m,3} = \frac{Q_l^2}{g \cdot D_s^3}$$  \hspace{1cm} (Eq. 2-4)
In equation (Eq. 2-2) ~ (Eq. 2-4), the first subscript ‘m’ refers to the modified Froude Number; the second ‘m’ refers to that the Froude Number is applied in the water mold and the last digital term refers to the different regions defined in Fig. 2-1 of the entire mold. $Q_{in}$ is the flow rate of the water injection, $d_{in}$ is the inner diameter of the cylindrical inlet pin, $H_{b}$ is the depth of the bath, $D_{h}$ is the hydraulic diameter, $D$ is the diameter of the vessel and $g$ is the gravity acceleration. The last term is defined as six times the ratio of volume, $V$, to surface area, $A$: (see Appendix)

$$D_{h} = 6 \cdot \frac{V}{A}.$$  \hspace{1cm} (Eq. 2-5)

The physical model was designed according to the calculations of the modified Froude number in different regions. Firstly, the modified Froude numbers were calculated for both the production sampler and the rectangular-shaped laboratory sampler. It was assumed that the inlet velocity for both the production sampler and laboratory sampler were 0.16 m/s and 0.3 m/s, respectively.\textsuperscript{14, 15} Then, the water model vessel was used to calculate the flow rate needed. In this work, the Froude number in Region II is considered to be important for the modeling of flow pattern. Therefore, the similarity criteria were applied to this region (Eq. 2-3). Moreover, flow rates other than the calculated value were also used in the experiments to investigate the flow patterns for different flow rates. Thus, a similarity of the modified Froude number in the physical model $Fr_{mm}$ is applied to calculate the real velocity at the inlet of the steel sampler.

Based on the calculations of the Froude number similarity between the production sampler, laboratory sampler and physical model vessels, the dimensions given in Table 2-1 were used in the construction of the physical models. The vessel was made of transparent acrylic resin. It should be noted that due to the vessel was scaled up, similarities with respect to other important dimensionless numbers such as the Reynolds number could not be fulfilled. However, it was judged to be more important to consider the modified Froude number, since it allows a better modeling of what is actually the fluid flow in the original sampler. The different flow rates used for PIV measurement are shown in Table 2-2.
Table 2-1 Setup of the water vessels

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Dimension [mm]</th>
<th>Inlet pipe diameter, ( d ) [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lollipop-shaped</td>
<td>( \Phi ) 200</td>
<td>( W ) 75</td>
</tr>
<tr>
<td>Rectangular-shaped</td>
<td>( L ) 150</td>
<td>( H ) 200 ( W ) 140</td>
</tr>
</tbody>
</table>

Table 2-2 Experimental trials for the lollipop-shaped water vessel

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Trials</th>
<th>( Q_w ) [L/min]</th>
<th>( F_r ) [-]</th>
<th>( Q_s ) [L/min]</th>
<th>( v_s ) [m/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lollipop-shaped</td>
<td>1</td>
<td>6.1</td>
<td>3.5E-05</td>
<td>0.083</td>
<td>0.049</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>8.3</td>
<td>6.5E-05</td>
<td>0.113</td>
<td>0.066</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>10.4</td>
<td>1.0E-04</td>
<td>0.141</td>
<td>0.083</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>12.3</td>
<td>1.4E-04</td>
<td>0.167</td>
<td>0.099</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>14.2</td>
<td>1.9E-04</td>
<td>0.193</td>
<td>0.114</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>16.3</td>
<td>2.5E-04</td>
<td>0.221</td>
<td>0.130</td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>18.3</td>
<td>3.1E-04</td>
<td>0.248</td>
<td>0.146</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>20.2</td>
<td>3.8E-04</td>
<td>0.274</td>
<td>0.162</td>
</tr>
<tr>
<td></td>
<td>9</td>
<td>22.1</td>
<td>4.6E-04</td>
<td>0.300</td>
<td>0.177</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>23.9</td>
<td>5.4E-04</td>
<td>0.324</td>
<td>0.191</td>
</tr>
<tr>
<td>Rectangular-shaped</td>
<td>1</td>
<td>6.6</td>
<td>4.89E-05</td>
<td>0.8</td>
<td>0.45</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>8.2</td>
<td>7.55E-05</td>
<td>0.9</td>
<td>0.55</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>10.3</td>
<td>1.19E-04</td>
<td>1.2</td>
<td>0.70</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>12.3</td>
<td>1.70E-04</td>
<td>1.4</td>
<td>0.83</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>14.4</td>
<td>2.33E-04</td>
<td>1.7</td>
<td>0.97</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>16.0</td>
<td>2.88E-04</td>
<td>1.8</td>
<td>1.08</td>
</tr>
</tbody>
</table>

\( Q_w \): Flow rates in the water vessel  
\( Q_s \): Flow rates in a real sampler  
\( v_s \): Corresponding velocity in a real sampler

2.2 Measurements of Flow Fields

Measurements of flow fields were carried out using a Twins Ultra PIV system. This system uses a Charge-Coupled Device (CCD) MEFAPLUS Camera with a speed of 30 fps to capture the flow containing small particles on a selected plane. Before carrying out the experiments, water was mixed with fluorescent seeding particles in order to capture the flow fields. The size of one particle is 15\( \mu \)m with a density of 1.1 g/cm\(^3\). When the plane is illuminated by two short duration laser flashes (green-colored laser, Nd:YAG), a double exposure of flow field is captured by the CCD camera and transmitted to the computer. Then, flow fields can be calculated by comparing the two photos through the PIV software. In this work, the middle plane of the vessel was illuminated by the laser flash at the right side. The CCD camera was set in front of the vessel, so that the front view of the flow patterns was obtained. A schematic plot of the whole measurement system is shown in Figure 2-2.
Fig. 2-2 Schematic plot of PIV system and experiment setup
Chapter 3

MATHEMATICAL MODELING

The mathematical model for the filling of a sampler is described in the follows.

3.1 Numerical Assumptions

The following assumptions were made in the mathematical modeling of the sampler fillings:

1. The flow rate during liquid steel extraction was constant;
2. All the air, water and molten steel were incompressible Newtonian fluids;
3. The properties of the solid and liquid phases were homogeneous and isotropic. In addition, the solid phase was stationary and rigid, and no micro-porosity forms;
4. The solid and liquid in the mushy zone were in local thermal and phase equilibrium. Furthermore, all thermophysical properties were constant except the steel density which was temperature dependent;
5. Heat transfer by radiation and convection from the free surface was negligible;
6. Only one quarter of the sampler (shown in Figure 3-1 as an example) was modeled due to the geometry symmetry in the computational domain for both the lollipop-shaped and rectangular-shaped sampler;
7. The inclusion particles were assumed to be spherical;
8. The particle collision, agglomeration and breakage were not considered.

3.2 Numerical Methods

The general form of the governing equation\textsuperscript{16} for the property $\Phi$ ($u, v, w, e, k, \omega$ and $T$) can be expressed as follows:

\[
\frac{\partial}{\partial t} (\rho \Phi) + \nabla \cdot (\rho \Phi \mathbf{u}) = \nabla \cdot (\Gamma \nabla \Phi) + S_{\Phi}
\]  

(Eq. 3-1)

where $\rho$ is the density, $\Phi$ is the general fluid property, $t$ is time, $\mathbf{u}$ is the mean velocity vector, $\Gamma$ is the diffusion coefficient and $S_{\Phi}$ is the source term. To represent the free surface shape of the liquid at the air/liquid interface, the VOF method created by Hirt and Nichols\textsuperscript{17} is used. In this method, the time dependent volume fraction $F$ is governed by the following equation,

\[
\frac{\partial F}{\partial t} + u_i \frac{\partial F}{\partial x_i} = 0. \quad (i = x, y, z)
\]  

(Eq. 3-2)
This equation states that $F$ moves with the fluid and is defined by the state of a control volume (cell) in the computational mesh:

$$F = \begin{cases} 
0 & \text{in the air phase} \\
0 < F < 1 & \text{at the interface} \\
1 & \text{in the liquid phase}
\end{cases} \quad \text{(Eq. 3-3)}$$

Due to the use of VOF model, physical properties are blended according to the percentages of volume fractions of each phase in the flow field. In this case, the physical property $\varphi$ can be expressed as follows:

$$\varphi = F \cdot \varphi_{\text{liquid}} + (1 - F) \cdot \varphi_{\text{air}} \quad \text{(Eq. 3-4)}$$

where $\varphi_{\text{air}}$ and $\varphi_{\text{liquid}}$ is the physical property of air and liquid, respectively. The different transport equations of property $\Phi$ in Equation (Eq. 3-1) are shown in Table 3-1.

Fig. 3-1 Schematic plot of the lollipop-shaped sampler computational domain
(left: top view, right: isometric view)

The movement of a single particle in the liquid steel was governed by the following equation:

$$\rho_p \frac{\pi}{6} d_p^3 \frac{du_p}{dt} = F_{\text{drag}} + F_{\text{buoy}} + F_{\text{grav}} \quad \text{(Eq. 3-5)}$$

where $\rho_p$ is the density of inclusion particle, $d_p$ is the diameter of the particle, $u_p$ is the particle velocity and $t$ is time. $F_{\text{drag}}$, $F_{\text{buoy}}$ and $F_{\text{grav}}$ are the drag force, buoyancy force and gravitational force, respectively.
Table 3-1 Conservation Equations

<table>
<thead>
<tr>
<th>Conservation of:</th>
<th>Φ</th>
<th>Γ</th>
<th>S_Φ</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass</td>
<td>l</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

- x-momentum
  \[ u = \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( \mu_e \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_e \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_e \frac{\partial u}{\partial z} \right) \]

- y-momentum
  \[ v = \frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left( \mu_e \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_e \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_e \frac{\partial v}{\partial z} \right) + \rho g \]

- z-momentum
  \[ w = \frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left( \mu_e \frac{\partial w}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_e \frac{\partial w}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_e \frac{\partial w}{\partial z} \right) \]

**k-ε model**

- Turbulence kinetic energy
  \[ k = \mu + \frac{\mu_t}{\sigma_k} \]

- Turbulence dissipation rate
  \[ ε = \mu + \frac{\mu_t}{\sigma_ε} \]

\[ \frac{\partial}{\partial t}(\rho \varepsilon) = \frac{\partial}{\partial x} \left( \mu + \frac{\mu_t}{\sigma_ε} \right) \frac{\partial \varepsilon}{\partial x} + \cdots - \frac{2}{3} \rho k \frac{\partial u_i}{\partial x_i} \delta_{ij} - \beta^* \rho \kappa \omega \]

\[ \frac{\partial}{\partial t} \left( \frac{k}{\rho} \right) = \frac{\partial}{\partial x} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x} + \cdots - 2 \rho \kappa \left( \frac{\partial u_i}{\partial x_i} \right) \delta_{ij} - \beta^* \rho \kappa \omega \]

**k-ω model**

- Turbulence kinetic energy
  \[ k = \mu + \frac{\mu_t}{\sigma_k} \]

- Specific dissipation rate
  \[ \omega = \frac{\mu}{\sigma_ω} \]

\[ \frac{\partial}{\partial t}(\rho \omega) = \frac{\partial}{\partial x} \left( \mu + \frac{\mu_t}{\sigma_ω} \right) \frac{\partial \omega}{\partial x} + \cdots - \frac{2}{3} \rho k \frac{\partial u_i}{\partial x_i} \delta_{ij} - \beta^* \rho \kappa \omega \]

\[ \frac{\partial}{\partial t} \left( \frac{k}{\rho} \right) = \frac{\partial}{\partial x} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x} + \cdots - 2 \rho \kappa \left( \frac{\partial u_i}{\partial x_i} \right) \delta_{ij} - \beta^* \rho \kappa \omega \]

Energy

\[ T = \frac{1}{C_p} \frac{\partial}{\partial t} \left( \frac{\rho}{C_p} \frac{\partial T}{\partial t} \right) \]

\[ \frac{\partial p}{\partial t} - \frac{\partial p}{\partial T} = \frac{\partial}{\partial x} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial T}{\partial x} + \cdots - \frac{2}{3} \rho k \frac{\partial u_i}{\partial x_i} \delta_{ij} - \beta^* \rho \kappa \omega \]

Notes:
- Laminar viscosity: \( \mu \);
- Turbulent viscosity: \( \mu_{k,ω} = \frac{\rho k}{\omega} \), \( \mu_{k,ε} = \frac{C_μ k^2}{ε} \)
- Turbulent intensity: \( \varepsilon \); \( \mu_e = \mu + \mu_t \);
- Heat capacity: \( C_p \);
- Effective thermal conductivity: \( \kappa \);
- Hydraulic diameter: \( D_h = 0.006 \) m;
- Turbulence kinetic energy: \( \frac{1}{2} (\frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_j}) \)
- Total pressure: \( \frac{\partial p}{\partial T} = \rho \frac{\partial}{\partial T} \)

\[ \frac{\partial}{\partial T} \left( \frac{\partial p}{\partial T} \right) = \frac{\partial}{\partial x} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial T}{\partial x} + \cdots - \frac{2}{3} \rho k \frac{\partial u_i}{\partial x_i} \delta_{ij} - \beta^* \rho \kappa \omega \]

Table 3-2 Inlet boundary conditions

<table>
<thead>
<tr>
<th>ν [m/s]</th>
<th>D_h [m]</th>
<th>Re</th>
<th>I [%]</th>
<th>k [m²/s²]</th>
<th>l [m]</th>
<th>ε [m²/s]</th>
<th>ω [s⁻¹]</th>
<th>μ_t [Pa·s]</th>
<th>μ/μ</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.40</td>
<td>0.006</td>
<td>2740</td>
<td>5.9</td>
<td>8.5E-04</td>
<td>4.2E-04</td>
<td>9.69E-03</td>
<td>11.4</td>
<td>0.51</td>
<td>85.1</td>
</tr>
</tbody>
</table>

3.3 Boundary Conditions

At the inlet, a velocity boundary was employed. The calculation methods of the employed boundary condition are summarized as follows and the calculation results are shown in Table 3-2:

- a) Velocity: \( u = 0.40 \) m/s;
- b) Turbulent intensity:
  \[ I = 0.16Re^{-0.125} \] (Eq. 3-6)
  where \( Re \) is the Reynolds number;
- c) Hydraulic diameter: \( D_h = 0.006 \) m;
- d) Turbulence kinetic energy:
\[ k = 1.5(u\cdot I)^2 \]  
(Eq. 3-7)

e) Turbulence length scale:
\[ l = 0.07D_h \]  
(Eq. 3-8)

f) Turbulence dissipation rate:
\[ \varepsilon = C_\mu \frac{k^{3/2}}{l} \]  
(Eq. 3-9)

g) Specific dissipation rate:
\[ \omega = C_\mu \frac{k^{1/2}}{l} \]  
(Eq. 3-10)

A small outlet on the top of the sample body was used in order to release the gas phase in the domain. This boundary is defined as pressure outlet boundary. In addition, a value of zero Pa was set to represent the gauge pressure.

3.4 Properties of Materials

For the two phases, air/water or air/liquid steel were employed in the simulation. Their material properties are summarized in Table 3-3. The property of liquid steel is assumed to be the stainless steel 304.

<table>
<thead>
<tr>
<th>Properties</th>
<th>Air</th>
<th>Water</th>
<th>Steel</th>
<th>Metal-mold</th>
<th>Sand</th>
<th>Cardboard</th>
<th>Splash</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density, ( \rho ) [kg\cdot m(^{-3})]</td>
<td>1.225</td>
<td>998.2</td>
<td>( \rho_{\text{steel}}^* )</td>
<td>7880</td>
<td>1600</td>
<td>689</td>
<td>310</td>
</tr>
<tr>
<td>Viscosity, ( \mu ) [Pa\cdot s]</td>
<td>1.7895E-5</td>
<td>1.003E-3</td>
<td>0.006</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Heat capacity, ( c_p ) [J\cdot kg(^{-1})\cdot K(^{-1})]</td>
<td>1006</td>
<td>-</td>
<td>700</td>
<td>700</td>
<td>1050</td>
<td>500</td>
<td>900</td>
</tr>
<tr>
<td>Thermal conductivity, ( \kappa ) [W\cdot m(^{-1})\cdot K(^{-1})]</td>
<td>0.0242</td>
<td>-</td>
<td>20</td>
<td>35</td>
<td>0.63</td>
<td>0.21</td>
<td>0.4</td>
</tr>
<tr>
<td>Melting heat, ( H ) [J\cdot kg(^{-1})]</td>
<td>-</td>
<td>-</td>
<td>290000</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Liquid temperature, ( T_l ) [K]</td>
<td>-</td>
<td>-</td>
<td>1727</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Solid temperature, ( T_s ) [K]</td>
<td>-</td>
<td>-</td>
<td>1673</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Contact angle on the wall in the air, ( \alpha ) [(^\circ)]</td>
<td>-</td>
<td>90</td>
<td>132</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Surface Tension, ( \sigma ) [N\cdot m(^{-1})]</td>
<td>-</td>
<td>0.073</td>
<td>1.6</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

*Note: the density of steel is temperature dependent

3.5 Methods of Solutions

The commercial CFD solver ANSYS\textsuperscript{®} 12.1 was employed to solve the governing equations. The mesh of one quarter of the geometry was carried out by GAMBIT\textsuperscript{®} 2.4.6, a preprocessor of the solver. All the cells were hexahedrons. The PRESTO discretization method was used for solving the pressure. Furthermore, the second-order discretization schemes were applied for calculating the momentum, energy, turbulence kinetic energy \((k)\), turbulence dissipation rate \((\varepsilon)\) and specific dissipation rate \((\omega)\).
Chapter 4

RESULTS

The results of the two kinds of sampler studies include both experimental studies and numerical calculations. Firstly, the physical modeling results are given and used to verify the mathematical model. Then, results from simulation of the filling of steel are presented including the consideration of initial solidification and inclusion dispersions.

4.1 Experimental Results of Physical Modeling

4.1.1 Lollipop-Shaped Steel Sampler

Figure 4-1 shows the mean flow pattern in the middle of a lollipop-shaped vessel at different filling times for an average flow rate of 20.2 L/min. The nozzle which connects the cylindrical inlet pin is located at the bottom center of the circular body. As can be seen from the plots, two vortexes are created on each side of the vertical flow. Moreover, these two vortexes are the two completely separated vortexes with the ends at the surface wall in the front and back of the vessel. The formation of vortexes is caused by the strong circulation flow after the injected flow approaching the liquid/gas surface. The flow pattern beneath the two vortexes clearly shows that water flows horizontally toward the center. As the bath depth increases, the centers of vortexes will also increase. Moreover, the size of the vortexes will enlarge in the end of filling.

Figure 4-2 depicts the development of flow pattern in the middle of the lollipop-shaped vessel at different filling times for a mean flow rate of 8.3 L/min. A comparison with the data presented in Fig. 4-1 shows that the filling flow rate is more than 50% less. Based on the data presented in Fig. 4-2 and other experiments data, some characteristic flow fields can be identified. Specifically, the following three regions were identified: vortexes, vertical flow field and horizontal flow field, as shown in the sketch in Figure 4-3. However, in the early stage of filling, circulations are formed near the bath surface. This is caused by the injection flow through the nozzle and also a small cross section area in the lower region of the sampler. As a result, a small hump is formed at the initial filling, but it is quickly immersed into the rising free surface. This observation is slightly different from what was seen in the flow pattern shown in Fig. 4-1, which shows a velocity field in the top of the vertical flow pattern due to the high flow rate. Centers of vortexes also shift from the region near the surface to a deeper region.
Fig. 4-1 Flow fields in the middle plane of a lollipop-shaped vessel ($Q_L = 20.2$ L/min)

Fig. 4-2 Flow fields development in the middle plane of a lollipop-shaped vessel ($Q_L = 8.3$ L/min)

Fig. 4-3 Schematic plot of flow pattern inside the lollipop-shaped vessel of initial filling (a), and finishing (b). (Region I: Vortex; Region II: Vertical flow field; Region III: Horizontal flow field)

Humps were observed since the process of filling a sampler can be considered as a typical bottom filling of an empty vessel with a constant fluid flow. It is known that the height of the
hump will decrease with the increased free surface. On the other hand, the measured maximum heights of the hump during each filling process increases with an increased flow rate. These results are in agreement with those obtained by Hallgren et al.\textsuperscript{21}, who have extensively studied the hump formation during filling of ingots. Figure 4-4 shows a linear relation between the maximum heights of hump and various flow rates. Their relation can be expressed as follows:

$$h = 3.11Q_L - 24.31 \quad (12 < Q_L < 22)$$

(Eq. 4-1)

where $h$ [mm] is the maximum height of a hump and $Q_L$ [L/min] is the flow rate.

Figure 4-4 shows a linear relation between the maximum heights of hump and various flow rates. Their relation can be expressed as follows:

$$h = 3.11Q_L - 24.31 \quad (12 < Q_L < 22)$$

(Eq. 4-1)

where $h$ [mm] is the maximum height of a hump and $Q_L$ [L/min] is the flow rate.

**Figure 4-5** shows the relationship between the ratios of the height of the vortex center to the height of free surface ($h/H$) at different flow rates. Because different flow rates lead to different filling times for the same amount of volume, all the data were obtained at the end of each filling process. As can be seen from the plot, the tendency is that the ratio decreases with an increased flow rate. This means that the height of vortex center increases slower than that of free surface. Therefore, the vortex region will expand into Region III (see Fig. 4-3) with an increased flow rate. This is because at the end of each filling process, the free surface area will decrease with the increasing height of free surface due to the circular cross section of the sampler vessel (above the center of the water vessel). The wave frequency on the surface was observed that it is increasing with an increasing height (above the center of water vessel, the area of surface will decrease according to an increasing height), approximately from 2 Hz at the center height of water vessel to about 2.5 Hz on the 80% of height of free surface at a flow rate of 18.3 L/min. A larger area in the flow is preferred by the vortex recirculation if the free surface area is small. Thus, the center of the vortex is switched into a larger area in Region III.
4.1.2 Rectangular-Shaped Steel Sampler

Figure 4-6 shows the mean velocity distributions in the middle of a sampler vessel at different filling times for an average flow rate of 8.2 L/min. The nozzle which connects the cylindrical inlet pin is located at the bottom center of the rectangular body. As can be seen from the plots, the flow fields can also be classified into three regions: i) the vertical flow region in the middle above the nozzle, ii) the vortex region near the free surface, and iii) the vertical flow region beneath the vortex region. The formation of vortexes is due to the strong recirculation created after the injected flow has reached the liquid/gas interface. Thereafter, it is directed towards the wall of the sampler vessel.

Figure 4-7 depicts the relationship between the heights of the vortex center to the height of the free surface at different flow rates. Data were obtained at the end of each filling process. As can be seen from the plot, the ratios for the rectangular vessel fall into a range of between 0.5 and 0.7 for all flow rates. However, it has a main tendency that the ratio will decrease with an increased flow rate. From the data in Fig. 4-7, the following equation can be derived:
\[ H_{\text{vortex center}} / H_{\text{free surface}} = -0.00825 \times Q_L + 0.74 \]  
(Eq. 4-2)

**Fig. 4-7** Ratio of height of vortex center to the height of free surface at different flow rates

**Fig. 4-8** Initial height of hump at different flow rates

Humps were observed during all the fillings. The initial height of the hump increases with an increased flow rate at the nozzle, as shown in **Figure 4-8**. It presents two groups of data gathered when measuring the initial height of the hump at three different flow rates. It is clear that the hump height will decrease with an increasing water surface height. From the data in Fig.4-8, the following equation can be derived for the initial hump height variation with the flow rate:

\[ H_{\text{hump}} = 3.7 \times Q_L - 26.3 \]  
(Eq. 4-3)

### 4.2 Water Flow Simulation and Mathematical Validation

#### 4.2.1 Lollipop-Shaped Steel Sampler

**Figure 4-9** shows the contours of the volume fraction of water on a middle plane of the computational domain at different filling times when using the two different turbulence models. As can be seen from the plots, very distinct flow pattern were obtained by employing the two different models. Clearly, the realizable \( k-\epsilon \) model shown in Fig. 4-9a can keep providing a very smooth interface between the gas and liquid during the filling process. However, a more realistic flow pattern on the free surface may be reflected by the predictions of a Wilcox \( k-\omega \) model as shown in Fig. 4-9b and compared with the data in Fig. 4-9c. The incoming flow from the inlet pin to the sampler body creates a hump on the free surface. It demonstrates a free surface without any fluctuations. In addition, no air is entrapped into the filling water calculated by the realizable \( k-\epsilon \) model; but some entrapped air can be found in the results of Wilcox \( k-\omega \) mode.
Fig. 4-9 Contours of volume fraction of water at different filling times by different models, flow rate $Q = 20.2\text{L/min}$

Fig. 4-10 Experimental and calculated velocity magnitude values in the middle plane of the sampler (where $Q = 8.3\text{ L/min}$. a. $H = 8.9\text{ cm}$; b. $H = 8.0\text{ cm}$)

Figure 4-10 depicts the comparison of velocity magnitude between the experimental results and the calculated results by different models at different heights from the inlet pin at
the bottom center of the sampler. It is seen that the flow has a higher velocity magnitude at the middle of sampler right above the inlet pin. This is due to a strong impact of flow directly released from the inlet pin entering the main sampler body. After reaching a certain height, the impact flow will collapse downwards and flow to the wall. Thus, the velocity of flow beside the middle vertical flow has a relative small magnitude. There is a fairly good agreement between the experimental result and calculated result predicted by the Wilcox $k-\omega$ turbulence model. Moreover, the mean deviation of the calculated result from the measurement is within 30% or even within 10% in some part of regions.

![Comparison between calculated velocity vectors and PIV measurement results (Q = 8.3 L/min)](image)

Fig. 4-11 Comparison between calculated velocity vectors and PIV measurement results (Q = 8.3 L/min)

The plots of velocity vectors calculated by the two different turbulence models and measured by PIV at different filling times are shown in Figure 4-11. It can be quickly identified that the Wilcox $k-\omega$ model predictions have a very similar flow pattern to those determined by the PIV measurements. More specifically, the very important locations of
vortex centers calculated by the Wilcox $k-\omega$ model show a very good agreement with the PIV results. Both results also indicate that the vortex centers are near the free surface beside the vertical flow region. Moreover, the flow field beneath the vortex region also shows a similarity with respect to both the calculated and measured results. However, the realizable $k-\varepsilon$ model shows a limited similarity to the velocity field compared to the PIV results. It overestimates the flow pattern beneath the free surface. As a result, the size of vortexes and their center positions are quite distinct from the PIV results. Specially, the vortexes region occupies almost the whole regions except the vertical flow region in the middle of the sampler body. On the other hand, it underestimates the fluctuations and hump formation on the free surface. Overall, the realizable $k-\varepsilon$ model has difficulties to simulate the flow field during this filling process. Although, some deviation of velocity magnitude shown in Fig. 4-10, the flow fields predicted by the Wilcox $k-\omega$ model show a very good agreement with the results obtained by PIV measurements.

\[ \text{Fig. 4-12 Ratio of height of vortex center to the height of free surface at different flow rates.} \]

\textbf{Figure 4-12} depicts the comparison of experimental and calculated results for the ratios of the height of vortex center to the height of free surface at different flow rates. The calculation results were obtained by using the Wilcox $k-\omega$ turbulence model. These data show that the predicted ratios agree well with the experimental results.

\textbf{4.2.2 Rectangular-Shaped Steel Sampler}

Velocity magnitude obtained by the PIV results and the calculation results at different height of a rectangular-shaped sampler vessel were compared. The results are shown in Figure 4-13. For the symmetrical geometry used in the simulations, only half of the calculation results are shown in the plot. In this case, the flow rate applied was 12.3 L/min. It can be seen that the flow has a higher velocity magnitude at the middle of the sampler, right above the inlet pin. The injected fluid will flow to the surrounded regions. The velocity
magnitude decrease significantly from the center to the wall. The agreements of the velocity magnitudes between the calculated and experimental results are fairly good.

![Graphs showing velocity magnitude decrease from center to wall](image1)

**Fig. 4-13** Experimental and calculated results in the middle plane of the sampler (Q = 12.3 L/min)

![Velocity fields verification](image2)

**Fig. 4-14** Velocity fields verification (Q = 12.3 L/min at filling time t = 8.7s)

The plot of velocity vectors fields predicted by PIV measurements and mathematical calculations are shown in **Figure 4-14**. The velocity fields agrees well between the two results. Both results depict that there is a recirculation vortex region, which is located near the free surface to the wall. In addition, the results also indicate that the vertical flow region above the nozzle is stronger than the other flow regions.

4.3 Simulation of Steel Flow in the Lollipop-Shaped Sampler without Solidification

It has been considered that the property of molten steel, such as composition of steel and the inclusion distributions, can be influenced by the flow patterns. This is true even after solidification. For example, in order to preserve the inclusions’ characteristics and their size distributions, a very calm filling is desirable. Therefore, it is necessary to study the flow pattern during its filling process.
Figure 4-15 shows the contours of liquid steel at different filling times. Here, it should be mentioned that the whole figure is plotted by mirroring x- and z-axis from the simplified symmetry calculation domain. Various inlet flow rates of 0.34, 0.51 and 0.68 L/min were applied to examine the flow pattern. As can be seen from the plots, the sampler is smoothly filled at a low filling rate (0.37 L/min), although the surface of liquid steel fluctuates somewhat. A hump is formed during the initial filling at a higher flow rate. And the surface wave is significantly larger than the flow pattern applied at a lower flow rate. The surface oscillations totally depend on the flow rate applied at the inlet. Comparing the different filling rate, it is seen that the higher flow rate applied, the larger oscillations appear on the surface. Thus, a bigger hump is created.

Figures 4-16 to 19 depict the mean flow properties at the different flow regions that are defined in Figure 4-3(a) during the fillings. Mean velocity magnitudes at the middle period of
filling for the three distinct flow regions are shown in Figure 4-16. The velocities at different regions were determined at the center part of each region. As can be seen from the plot, the velocity magnitudes at the vertical flow region in Region II are the highest among the three regions. The average velocity during the filling is 0.4 m/s in this region. The mean velocity magnitude of Region III is 0.1 m/s, which is only 25% of the magnitude in the vertical flow region. However, the vortex region which is usually located beneath the free surface has an average velocity magnitude of 0.016 m/s, which is 96% less than that in the vertical flow Region I. It can also be seen that the divergence from the mean velocity in the same flow region at different filling times is small.

Fig. 4-16 Mean velocity magnitude at different flow regions during filling (Q = 0.5 L/min)

Fig. 4-17 Mean turbulence kinetic energy at different flow regions during filling (Q = 0.5 L/min)

The mean turbulence kinetic energy \((k)\) at different flow regions during filling are expressed in Figure 4-17. Different flow fields have distinct magnitude of turbulence kinetic energy during the fillings. It is found that it is much higher in the vortex region than that in the other two flow regions, especially in the beginning of the filling process. It is also noticed that the mean velocity magnitude in this flow region is the lowest among the three regions as shown in Fig. 4-16. However, the turbulence kinetic energy is defined by the velocity fluctuations\(^\text{16}\), which can be expressed as follows:

\[
k = \frac{1}{2} \left( \overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right)
\]  
(Eq. 4-4)

where \(\overline{u'}\), \(\overline{v'}\) and \(\overline{w'}\) are the velocity fluctuations in the \(x\), \(y\) and \(z\) directions. Thus, it is found that although the velocity magnitude in the vortex region is very low, the turbulence kinetic energy is very high. This is due to the strong velocity fluctuations in this region. Moreover, it is also seen that there is a sharp decrease of \(k\) during the middle period of fillings. This may be because of the initial chaotic flow pattern, which is caused by the vortex formation. In addition, the velocity fluctuations are also due to the surface fluctuations.
The specific dissipation rate (\( \omega \)) and the turbulence intensity (\( I \)) calculated from the Wilcox \( k-\omega \) turbulence model are illustrated in Figure 4-18 & 19, respectively. The specific dissipation rate decreases for the flows in Region I & III, but it increases in the flows in Region II. The trends of turbulence intensity in Fig. 4-19 are found to be very similar to those found for the turbulence kinetic energy in Fig. 4-17.

![Figure 4-18](image1.png) Mean specific dissipation rate at different flow regions during filling (Q = 0.5 L/min)

![Figure 4-19](image2.png) Mean turbulence intensity at different flow regions during filling (Q = 0.5 L/min)

**Figure 4-20** presents the relationship between the ratio of the height of the vortex center (at the middle plane of the sampler from the front view) to the height of free surface and the filling times. The purpose of defining this ratio is to compare the vertical positions of the vortex center in the up-raising fluid flow. From this figure, it is seen that all the ratios are around 0.8 through all filling times. Generally speaking, the slightly higher the inlet flow rate applied, the higher the ratio will be. Accordingly, the vortex center will be closer to the free surface. Moreover, the data in the plot also shows that the ratio is not constant for a specific inlet flow rate.

![Figure 4-20](image3.png) Ratio of height of vortex centers to the height of free surface at different filling times.

26
4.4 Simulation of Steel Flow with Initial Solidification in the Lollipop-Shaped Mold

The solidification applied in this work aims at modeling the inclusion particles’ dispersions in a more realistic manner. In this study, a simple solidification model was used without taking into account any phenomena such as the shrinkage, pores’ formation, segregation and air gap between the metal and mold, etc. It also assumes that no temperature gap exists between different materials.

Due to the specific geometry of the sampler mold design (shown in Fig. 1-2), unique contours of steel solidification profiles are shown in Figure 4-21. The plots show that the solidification starts shortly after contacting with the metal mold. The solidification rate is faster in the sample thickness direction than in the width direction. The sample surface at the bottom solidifies first followed by the upper surface and the inner body.

![Figure 4-21 Contours of solidified steel fraction during sampling (1: liquid 0: Solid)](image)

![Figure 4-22 Comparison of velocity magnitude at a specific position (shown as the dotted part) between the liquid steel filling of a sampler with and without solidification](image)

![Figure 4-23 Calculated solidified shell thickness at the top of inlet pin](image)

The maximum velocity magnitude at the connection area between the main mold and the inlet pin is shown in Figure 4-22. A homogenous velocity profile was applied at the velocity inlet with a magnitude of 0.4 m/s. Due to the turbulence model and the presence of the
viscous layer at the wall, the homogenous velocity profile at the inlet will develop into a turbulent flow pattern in the end of inlet pin. Thus, the velocity magnitude increases. As can be seen, the velocity keeps steady in the case without considering solidification. However, due to the solidification of the liquid steel at the inner wall of metal mold, the inner diameter of the inlet pin shrinks. According to the Bernoulli’s principle of flow in a pipe, the velocity magnitude is inversely to the area of the cross section for a constant flow rate. The liquid steel starts to solidify immediately when it is in contact with the metal mold. Thus, the flow of the molten steel will be accelerated, leading to a significant increase of velocity magnitude. Furthermore, the thickness of the solidified shell varying with time during the filling of a sampler is shown in Figure 4-23, which can be applied to verify the velocity decrease in Fig. 4-22.

A dramatic increase of the solidified shell thickness at the beginning of the filling is seen in the plot. The molten steel solidifies immediately when it is in contact with the mold. As soon as the metal mold begins to heat up, the rate of heat transfer from the melt to the mold will decrease. Being heated by the super-heated melt, parts of the already solidified steel at the solid/liquid interface will be re-melted again. Thus, it is noticed that the thickness of the solidified shell decreases gradually during the following filling times. Thereafter, the velocity will also decrease according to the enlarged channel in the inlet pin. Just before the sampler is totally filled up with steel, some numerical coding has been applied to gradually decrease the filling velocity at the velocity inlet. It aimed at decreasing the filling velocity to zero when the sampler cavity is just entirely filled with steel. From a numerical point of view, it can also assist to get a better convergence of the calculation. The flow rate of steel will also decrease gradually according to the decreased filling velocity. As can be seen from Fig. 4-23, the pre-solidified shell thickness suspends at about 1 s because of the low flow rate of liquid steel, which leads to a low flow rate of incoming melt that carries the re-melted steel away into mold. Also, the amount of heat applied to the solidified shell decreases. Until the incoming flow of melt cannot take the re-melted steel away into the main mold, the thickness of the shell increases. When the filling ceases, no more melt is filled into the mold. Thus, the heat transfer is only in the direction from the melt to the metal mold, which leads to a strongly increased thickness of the shell.

The liquid fractions of steel at different filling times from center to the surface are shown in Figure 4-24. As the results of the influence of inhomogeneous heat transfer from melt to the mold at initial filling and especially the turbulence during filling, the liquid fraction on the center line is disturbed. As soon as the filling ceased after one second, flow inside the mold damps with time. Therefore, the heat transfer and solidification are not influenced by turbulence anymore. A very clear solidification front can be distinguished as shown in the plot. Thereafter, solidification will continue until all the liquid is transformed into solid.

Figure 4-25 demonstrates the temperature verification at the center point of a sample. Both calculated and experimental results are shown. The experimental measurements have been repeated several times. The mean filling velocity of the sampler during the experiments is between 0.34 and 0.71 m/s. However, due to the uncertainty of industrial plant trials at a
very high temperature, the results differ somewhat among the four trials. The calculated result depicts that the temperature is kept above the liquidus during the filling within the first second. Then, it starts to decrease at 0.75 seconds. More specifically, the solidification starts at 0.95 second with an average solidification rate of 40 °C/s. During the solidification from 0.95 to 2.3 seconds, the agreements between predicted and experimental results are good. After the first 2.3 seconds, the solidification at the center point was completed. The cooling curve can be characterized into three regions: I. Cooling of melt; II. Solidification; III. Cooling of solidified metal. Three parameters determine the shape of the cooling curve: the heat capacity of both superheated melt and the solidified metal, and the heat of fusion of the metal. The released heat of fusion keeps the temperature roughly constant in the first period of solidification. The solidification rate increases a lot during the last filling period. Besides, due to the limitation of this simple solidification model, a dramatic temperature decrease occurs immediately after the solidification.

4.5 Dispersions of Inclusions

It is important to understand the inclusions dispersions in the steel sample after solidification. Here, the concentrations of particles are calculated within the steel sample. Five different sized particles were applied during the calculation. According to the results of three dimensional investigations of inclusions in stainless steel samples, the peak of the inclusions size distribution corresponds to a size of about 1.3 µm in the experiments. Therefore, the choice of tracked particles was based on the experimental results and the sizes were 0.8, 1.3, 2, 5 and 10 µm, respectively.

According to Equation Eq. 3-5, three different forces were considered to exert on the inclusions in the liquid steel flow. Table 4-1 presents the calculated values of forces for different sized inclusions in different regions (shown in Figure 4-26). As can be seen in the table, the applied drag force is much higher than both of the buoyancy and gravitational force.
that is exerted on the same inclusion. In addition, both the buoyancy and gravitational forces increase with an increasing size of inclusions. Fig. 4-26(a) is the center of a sample in a front view. It also characterizes into three different flow regions as defined previously. In Fig. 4-26(b), the regions are chosen as the center of vortex flow region, and both inner and outer vortex flow regions. Another important flow region is at the velocity inlet. The calculated results show that the drag force for a single inclusion is the highest in the inlet flow region. This is due to the assumption of zero velocity of inclusions at the inlet. Therefore, the drag forces need to accelerate the motion of inclusions. The low drag force in both Region I & II is because the velocity difference between the fluid and the inclusion is very small. It is found that the drag force of an individual inclusion in Region IV is relatively higher than those found in regions V & VI. This may be because the recent injected inclusions from inlet pin with the incoming fluid flow need to be decelerated significantly as soon as flowing into the vortex region. Therefore, the inclusions may concentrate in this region. From this table it can also be concluded that the movement of inclusions is mainly governed by the fluid drag force.

**Fig. 4-26** Positions in the sample where the forces exerting on inclusions are calculated. (a. 0.68 s; b. 0.96 s)

**Table 4-1** Calculated values of forces exerting on inclusions at different regions

<table>
<thead>
<tr>
<th>$d_p$ [µm]</th>
<th>Inlet region</th>
<th>Region I</th>
<th>Region II</th>
<th>Region III</th>
<th>Region IV</th>
<th>Region V</th>
<th>Region VI</th>
<th>$F_{drag}$ [N]</th>
<th>$F_{buoy}$ [N]</th>
<th>$F_{grav}$ [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.8</td>
<td>1.8E-08</td>
<td>4.4E-10</td>
<td>3.3E-10</td>
<td>2.2E-09</td>
<td>1.1E-08</td>
<td>5.2E-09</td>
<td>1.9E-09</td>
<td>1.8E-14</td>
<td>7.9E-15</td>
<td></td>
</tr>
<tr>
<td>1.3</td>
<td>3.0E-08</td>
<td>7.2E-10</td>
<td>5.3E-10</td>
<td>3.6E-09</td>
<td>1.8E-08</td>
<td>8.4E-09</td>
<td>3.2E-09</td>
<td>7.7E-14</td>
<td>3.4E-14</td>
<td></td>
</tr>
<tr>
<td>2.0</td>
<td>4.9E-08</td>
<td>1.1E-09</td>
<td>8.1E-10</td>
<td>5.5E-09</td>
<td>2.8E-08</td>
<td>1.3E-08</td>
<td>4.9E-09</td>
<td>2.8E-13</td>
<td>1.2E-13</td>
<td></td>
</tr>
<tr>
<td>5.0</td>
<td>1.4E-07</td>
<td>2.8E-09</td>
<td>2.0E-09</td>
<td>1.4E-08</td>
<td>7.8E-08</td>
<td>3.4E-08</td>
<td>1.2E-08</td>
<td>4.4E-12</td>
<td>1.9E-12</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>3.2E-07</td>
<td>5.5E-09</td>
<td>4.1E-09</td>
<td>2.9E-08</td>
<td>1.7E-07</td>
<td>7.5E-08</td>
<td>2.5E-08</td>
<td>3.5E-11</td>
<td>1.5E-11</td>
<td></td>
</tr>
</tbody>
</table>

Due to the calculation limitation (hardware), the total number of 384,000 tracked particles for simulation is significantly less compared to the real number of inclusions that are present in a production steel sample. But the amount of tracked particles in this work is enough for the analysis and comparison of the main tendencies for distribution of inclusions with different sizes in the steel sample. Besides, as the particles will disperse all over the calculation domains of the fluid, some numerical code have been developed to constrain the movement of inclusions within the steel phase domain only. The concentrations of dispersed
particles in a solidified sample were analyzed in different zones and different parts, as illustrated in Figure 4-27. Each analyzed zone has the same volume with a dimension of $12 \times 6 \times 0.2$ mm$^3$. The initial concentration of inclusions in liquid steel for each sized particle is $6.22 \text{ mm}^{-3}$.

**Figure 4-27** Different zones and parts of analyzed Lollipop-shaped samples

**Figure 4-28** Experimental results of inclusion density in different zones on surface of a Lollipop-shaped sample

Figure 4-28 shows the experimental results obtained from a 3D investigation of particle size distributions in different zones on the surface of a 12 mm thickness lollipop-shaped stainless steel sample that taken during the ladle treatment. It was found that the top zone contains the highest concentration of inclusions. Slightly more particles are found in the bottom zone than in the middle. The ratio between the total number of inclusions in the top and middle zones is around 1.7. The calculated results shown in Figure 4-29 also depict the same tendency as described in Fig. 4-28 and verified by it. The mean total inclusion number ratio is about 2, which is quite similar to the experimental results. However, as shown in Fig. 4-29, the predicted inclusion concentrations in different zones of sample center show a quite different trend. The calculation results in the center of sample show that more inclusion particles concentrate in the middle region of the sample than at center. The inclusions concentration in the middle of the center area is about three times larger than that found in the middle of the surface area. This may be due to the vortex formed by turbulence, which can bring the inclusions into the recirculation flow causing a high concentration of the inclusions. In addition, this recirculation flow with a high turbulence energy dissipation rate can also lead to a high particle collision rate. On the other hand, the inclusion dispersions on the top and bottom zones in the center area are almost the same. However, it should be noted that a more advanced solidification modeling might have resulted in different results.
From the above shown plots it is seen that the concentrations of different sized inclusions on sample surface are very low at both bottom and middle zones. In contrast, the inclusions concentrations at both the 1/2 center and the center are quite high for the middle region. The quite big difference is due to the high turbulence during the sampler filling. The recirculation flow in the vortex region tends to pull the inclusions into its centre. Thus, a higher concentration of inclusions is kept in the center of the vortex. This, in turn, will lead to more opportunities for the inclusions to collide and agglomerate for an industrial case. The first solidified region is the bottom parts next to the wall. The rate of solidification can be as high as 400 °C/s. Thus, the molten steel with fewer inclusions solidifies in these regions, causing a low concentration of inclusions. Because of the high turbulence during sampler filling, an inhomogeneous dispersion of inclusions in the entire sample can hardly be avoided.
each concentration zone in different groups are very similar. These plots reveal a very clear contour that most inclusions with varied sizes concentrate in the center area of a sample. The concentration of inclusions decreases gradually from the center to the periphery of a sample. It is also shown that inclusion concentrations of zone D in the sample center is about 3 times higher than that in zone A which is at the sample surface. Concentrations in zone B (between 5 mm$^{-3}$ and 7 mm$^{-3}$) are quite similar to the initial particles concentrations (6.22 mm$^{-3}$) in the liquid steel. Therefore, zone B can be employed as a proper zone of this Lollipop-shaped 12mm thickness sample for analyses of inclusion distributions after sampling. Other zones with either lower or higher concentrations compared to the initial value are not suitable for inclusion determinations.

The assumed concentration of inclusions (6.22 mm$^{-3}$) is significantly less than that the real concentration of inclusions (up to $5 \times 10^4$ mm$^{-3}$) in a production sample. The relation between the experimental and calculated results can be generally expressed as:

$$N_{\text{v-exp}} = C \cdot N_{\text{v-cal}}.$$  \hspace{1cm} (Eq. 4-5)

where $N_{\text{v-exp}}$ and $N_{\text{v-cal}}$ are the inclusions concentrations obtained by experimental and calculated results, respectively. The constant $C$ is the relation coefficient between them. The equation has been applied to different zones and the coefficient $C$ is evaluated by calculating the average value of the top, middle and bottom regions. Table 4-2 shows the results. It shows that the deviation of $C$ is smaller in the smaller sized inclusion range. At the range of $5.0 \sim 10.0 \mu m$, the deviation is as high as about 40%. However, the most of the inclusions are in a range of $0.8 \sim 2.0 \mu m$ for a production sample. It is seen that the calculated $C$ for different zones in this range is the least deviation. It demonstrates that the calculation results agree best with the experimental results in this range.

<table>
<thead>
<tr>
<th>Range [µm]</th>
<th>C [$\times 10^4$]</th>
<th>St. Deviation</th>
<th>%</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.8 ~ 2.0</td>
<td>0.94</td>
<td>0.12</td>
<td>13.0</td>
<td>$N_{\text{v-exp}} = (0.94 \pm 0.12) \times 10^4 \cdot N_{\text{v-cal}}$</td>
</tr>
<tr>
<td>2.0 ~ 5.0</td>
<td>1.07</td>
<td>0.24</td>
<td>22.9</td>
<td>$N_{\text{v-exp}} = (1.07 \pm 0.24) \times 10^4 \cdot N_{\text{v-cal}}$</td>
</tr>
<tr>
<td>5.0 ~ 10.0</td>
<td>1.11</td>
<td>0.45</td>
<td>40.5</td>
<td>$N_{\text{v-exp}} = (1.11 \pm 0.45) \times 10^4 \cdot N_{\text{v-cal}}$</td>
</tr>
<tr>
<td>Average</td>
<td>1.03</td>
<td>0.25</td>
<td>24.3</td>
<td>$N_{\text{v-exp}} = (1.03 \pm 0.25) \times 10^4 \cdot N_{\text{v-cal}}$</td>
</tr>
</tbody>
</table>

The predicted inclusion densities in the rectangular-shaped sampler are shown in Figure 4-31 for different zones (top, middle and bottom) from the sample surface to the center. It can be seen that the inclusion density in the surface region is much higher than that in the ½ center and center regions. The density on the surface region is about 5 times more than that on the center region. The high inclusion concentration on the surface region may be because of the vortex formation near the surface during filling. The recirculation flow will lead to capturing of the inclusions at the vortex center, which will cause a higher concentration of inclusions. Moreover, the tendency on the surface is that the top zone contains more inclusion particles than both the middle and bottom zones. The same tendency has been also observed and
predicted for other different steel sample geometries. On the ½ center region, the inclusion density at the top and bottom zones are higher than the middle zone. On the other hand, the density at the center region are quite similar for the three vertical zones. But the density is the least comparing to the entire sample body.

Since the inclusion density in the analyzed regions vary somewhat from the initial density, it can be predicted from the Fig. 4-31 that the region between the surface and the ½ center may have a very similar inclusion density as the initial value. It is important to know the correct positions for the determination of inclusions which has the same inclusion density as the original concentrations. Thus, if applying a inclusion determination for this particular sampler type, the region between the sample surface and ½ center should be used.

**Fig. 4-31** Predicted inclusion densities in different zones from the surface to center of a sample
Chapter 5

DISCUSSION

As mentioned previously in the introduction chapter, the steel sampler study is based on the two different shaped models: a lollipop-shaped sampler and a rectangular-shaped sampler that described in Fig. 1-3. The study of the two models is parallel. The study of the filling process of the lollipop-shaped production sampler was divided into four individual supplements with comprehensive results and discussions. However, for the rectangular-shaped sampler study, the same analytical methods were employed. All the physical modeling, mathematical modeling, validations and predictions are described within one supplement, Supplement V.

In this study, all the numerical simulations have been carried out by assuming a completely symmetrical calculation domain. Erroneously, these predictions neglect the fluctuating nature of a turbulent flow. Due to the fluctuations, even some very small changes of flow variables may promote radical changes of fluid flow patterns in the metallurgical vessels. The non-symmetry flow pattern can also be observed during the PIV measurements, for example, the velocity vectors of flow pattern shown in Supplement I. However, by using the half or even a fourth of the computational domain, the calculation time can be significantly reduced and the predicted results are still acceptable. Many other researchers employ the symmetrical property of geometries to carry out numerical studies, even though the flow pattern of turbulence is non-symmetrical. So do the current studies.

It is known that complex phenomena take place during the filling of a liquid steel sampler. It is difficult to simulate this process in a laboratory scale experiment. The difficulty of this dynamic process lies in the following aspects:

1. High temperature: The working temperature is at the steelmaking temperature above 1550 °C. During the sampling, the splash protection surrounded at the end of a sampler rod will burn when submerged into the melt. Also, the chemical reaction between the splash protection and the melt at an extremely high temperature will release quite amount of carbon dioxide, which will flow out to the surface of the melt. The large amount of floating carbon dioxide bubbles will lead to a significant movement of the sampler rod, causing very unstable sampling conditions;
2. Rapid solidification: The liquid steel solidifies immediately after it is sucked into the mold when it comes in contact with the wall. This has also been predicted and the results are shown in Supplement IV. Usually, shrinkage is found in the top
center of the solidified sample. Sometimes, porosities are found in the steel samples. On the other hand, a thin air gap will form between the solidified sample and the mold;

3. Reoxidation: Molten steel may oxidize in the metal mold, forming exogenous inclusions that originally did not exist in the melt;

4. Turbulence: The fluid flow is turbulent mainly because of the injection of steel from the inlet pin into the main mold. Besides, the shaking sampler rod will cause a more chaotic flow in the sampler mold. However, it is difficult to verify the turbulence if it is coupled with solidification;

5. Small scale: The dimension of the sampler mold is very small.

Thus, some simplification and assumptions were made during the numerical calculations. Such as the simplified solidification modeling used assumes no shrinkage, no pore formation, a good contact between each solid zones and no air gap between the solidified steel sample and the mold. Besides, the Wilcox k-ε model was used for the entire simulations of fluid flow of the sampling system. This turbulence model that has been verified works very well for the water modeling in Supplement II and the liquid steel simulations which do not consider solidifications in Supplement III. Due to the changed cavity of the mold after an initial solidification, the flow pattern will be influenced a lot. This is described in detail in Supplement IV. In this case, it is difficult to verify that if this turbulence model agrees best with the experimentally determined instantaneously changed flow patterns along with the narrow fluid flow space. Moreover, the inclusion model in both Supplement IV & V is also simplified. Only three forces were considered to influence the particles movements. In addition, the initial concentrations of inclusions were also assumed as well as the particle sizes. So the results of the inclusion tracking will only be the particle concentrations. Although lots of simplifications and assumptions were made to carry out the calculations, the tendencies of the numerical results agrees very well with the industry plant trials as seen in Fig. 4-28 & 29.

Generally speaking, it is always desirable to have an ideal filling condition with a very calm flow pattern and unchanged concentration of inclusions. So the initial inclusion size distributions in the ladles can be preserved to a large extent. However, after the investigation of the results, it is found that it is difficult to obtain such a condition. The turbulence presence during filling with an extremely chaotic flow pattern makes it is very difficult to maintain any accurate inclusion size distributions. For example, the calculated Reynolds number at the inlet pin areas described in Supplement III is around 2100 ~ 3500. This is just slightly more than the critical value between laminar and turbulent flow which is \( Re = 2100 \). However, due to the initial solidification at the inlet of the main mold, the inner diameter of the pin shrinks. In this case, the Reynolds number calculated in Supplement IV can reach values as high as 6000. Moreover, the burn of splash protection layer in practice will also leads to a release of considerable amounts of gas, which will cause a violent shaking of the sampling rod when the gas flows out. This, in turn, will result the steel flow to be even more unstable. Therefore, the inclusion density in the sampler mold will be changed from the initial uniform concentrations, which exists at the sample inlet.
Inclusions are believed to collide and even grow into bigger sized ones in a turbulent flow. The changed particles size distribution will lead to a wrong prediction of the conditions during the steelmaking. Previously, M. Söder et al.\textsuperscript{24} have theoretically evaluated the different inclusion growth mechanisms in an inductively-stirred ladle. It has been found that the turbulent collisions are much more important for the inclusion growth than the other growth mechanisms such as laminar collision and stokes growth. Furthermore, when the turbulence increases, the inclusions will more readily collide.\textsuperscript{24} More details are illustrated in Supplement III.

The colliding of two inclusions with a diameter of $r_i$ and $r_j$ in a turbulent flow can be calculated using the following collision volume $W_{ij}$ [m$^3$·s$^{-1}$] equation\textsuperscript{24}:

$$W_{ij} = 1.3\pi^2\alpha(r_i + r_j)^3 \sqrt{\frac{\varepsilon}{\nu_{Fe}}}$$

(Eq. 5-1)

where $\alpha$ is the collision efficiency, $\varepsilon$ is the turbulence dissipation rate and $\nu_{Fe}$ is the kinematic viscosity of the liquid steel. The empirical coefficient of collision efficiency, $\alpha$, was estimated to be 0.27 $\sim$ 0.63 by Nakanishi and Szekely\textsuperscript{25} by employing the inclusions with a density of 3000 kg·m$^{-3}$ in a liquid steel with a density of 7200 kg·m$^{-3}$.

The relation among the turbulence dissipation rate $\varepsilon$, turbulence kinetic energy $k$ and the specific dissipation rate $\omega$ can be expressed by the following equation:

$$\omega = \frac{\varepsilon}{k}$$

(Eq. 5-2)

Thus, the collision volume in the Wilcox $k$-$\omega$ turbulence model can be illustrated as:

$$W_{ij} = 1.3\pi^2\alpha(r_i + r_j)^3 \sqrt{\frac{\omega k}{\nu_{Fe}}}$$

(Eq. 5-3)

For collisions due to turbulence, the turbulence energy dissipation rate is vital. It is found to be high in two regions in the sampler during fillings: the vortex flow region and the free surface region. Also, it is much higher in the vortex flow region than in the other two flow regions as can be seen from Fig. 4-17 & 18. However, it is lower than in the free surface region as shown in Figure 5-1. But by comparing the maximum values in a gas-stirred ladle, which have been calculated by Jönsson and Jönsson,\textsuperscript{26} the turbulence dissipation rate in the sampler is much higher than that in the gas-stirred ladle. As is shown in the plot, the turbulence dissipation rate is assumed to be constant considering a steady state flow inside the ladle furnace. The maximum turbulence dissipation rate in the ladle for a gas flow rate of 80, 160 and 240 L/min is 0.175, 0.475 and 1.35 m$^2$·s$^{-3}$, respectively. The maximum dissipation rate in the sampler is 8.9 and 11.5 m$^2$·s$^{-3}$ in the vortex flow region and the free surface, respectively. This is much higher than that is found in a ladle.

Moreover, M. Söder et al.\textsuperscript{24} also predicted that the average energy dissipation rate is $8.73 \times 10^{-3}$ m$^2$·s$^{-3}$ in an inductively stirred-ladle with a 700 A stirring current. Another reported value by Tozawa et al.\textsuperscript{27} is $10 \times 10^{-3}$ m$^2$·s$^{-3}$ for an ASEA-SKF ladle. Thus, it can be
concluded that for the vortex region and the free surface in the sampler, the average turbulence energy dissipation rates are about 1,000 times higher than those in ladles with induction stirring. This in turn, will make the likelihood for growth due to turbulent collisions in samplers higher than in ladles.

![Fig. 5-1](image1.png) **Fig. 5-1** Comparison of turbulence dissipation rate between the values in the sampler (without solidification) and the value in the ladle furnace during steelmakings.

![Fig. 5-2](image2.png) **Fig. 5-2** Calculated collision volumes in the sampler (without solidification) and ladles \((d_p = 1.3 \, \mu m)\)

![Fig. 5-3](image3.png) **Fig. 5-3** Comparison of turbulence kinetic energy at the center of vortex region on the side of a sample. (inlet velocity \(v = 0.4 \, \text{m/s}\))

In **Figure 5-2**, the calculated collision volumes for two inclusion particles with the same diameter of 1.3 \(\mu m\) that collide with each other at different flow fields in the sampler are plotted. The same calculation was also carried out for a ladle using the average turbulence energy dissipation rate. The value of the collision efficiency, \(\alpha\), was chosen as 0.5. It is clearly seen from the plot that the collision volume in the sampler is much higher than that in the ladle. The average collision volume in the sampler without solidification is about 30 times higher than that in the ladle furnace. Within the sampler body, the high turbulence energy dissipation rates concentrate at the free surface. Thus, the collision volumes in this region are
higher than in the other regions. Due to the high turbulence energy dissipation rate in the steel sampler during its filling and also from the simulation results shown above, it is evident that

\[ W_{ij,\text{ sampler}} \gg W_{ij,\text{ ladle}} \]  

(Eq. 5-4)

where \( W_{ij,\text{ ladle}} \) and \( W_{ij,\text{ sampler}} \) is the collision volume in the ladle furnace and the liquid steel sampler, respectively. Therefore, the concentration of smaller sized inclusions will be decreased in the sampler mold, while the concentrations of bigger sized ones are increased. For example, Huet et al.\textsuperscript{28} found a higher concentration of larger inclusions in the upper part of the sampler body than in the lower part of the sampler body.

During the filling of a rectangular-shaped sampler, the average collision volume for the entire sampler mold is \( 1.92 \times 10^{-14} \text{ m}^3\cdot\text{s}^{-1} \) (assuming the two inclusion size are both 2 µm and \( \alpha = 0.5 \)). The collision volume in the vortex region is about \( 4.86 \times 10^{-14} \text{ m}^3\cdot\text{s}^{-1} \), which is about 2.5 times higher than the average value. However, the highest collision volume is found in the region surrounding the inlet at the bottom of mold with a value of \( 2 \times 10^{-13} \text{ m}^3\cdot\text{s}^{-1} \).

The calculated turbulence kinetic energy based on the velocity vectors shown above is plotted in Figure 5-3. The choice of the positions for the comparison is in the centers of the vortexes on the side of a sample. Both of these cases correspond to an inlet velocity magnitude of 0.4 m/s. As shown above, the height of the vortex centers increase with an increased free surface height. It is also seen from the plot that the turbulence kinetic energy in the case considering solidification is much larger than a case neglecting solidification. The turbulence kinetic energy decreases continuously until the end of fillings. Because of the free surface area decreases at the top of a sampler mold, the frequency of the wave on surface becomes higher. This may result in a large turbulence disturbance, and thus leads to a higher turbulence kinetic energy. After the completion of the filling, the flow in the sampler is damped with time. Therefore, the turbulence dissipates away, which causes a decrease of the kinetic energy.

Based on the results in Fig. 5-3, the relation of collision volume in both the case with and without solidification is:

\[ \frac{W_{ij}(\text{with solidification})}{W_{ij}(\text{without solidification})} = 1.8 \]  

(Eq. 5-5)

It illustrates that it is about 1.8 times more likely for the turbulent collisions to take place in the sample when considering solidification than that without taking solidification into account. Besides, it is also estimated in Supplement IV that the average collision volume in a production sampler is about 60 times higher than that in a ladle furnace (which is about \( W_{ij} = 1.1 \times 10^{-15} \text{ m}^3\text{/s} \)) when considering the steel solidification. Thus, the inclusion concentrations in the vortex regions during sampler fillings will be significantly changed from an initial density of inclusions in the ladle.
The calculated results of inclusion concentrations in different zones of the 12 mm thickness lollipop-shaped sample can be employed to determine the optimal position of the steel sample for inclusion analysis in the industry. These are also described in detail in *Supplement IV*. Results showed that for some calculated regions the predictions shows a good agreement with the initial conditions. However, for other regions this is not true. As can be seen from Fig. 4-30, the deviation is high between the initial concentration and the calculated results at the middle of the sample. Similarly, the calculated results at the bottom surface area deviates from the initial concentration a lot. The average deviation for different sized inclusion at these positions is about 50%. However, the calculations illustrate that the results on top and bottom zones except bottom surface area have approximately the same results as the initial concentrations. The mean deviation for all kinds of sized particles is within 10%. But the result is only from a simple numerical solidification model. From a metallurgical point of view, the sampler always has porosity and shrinkages in the top zone of a steel sample. In this case, the only proper position that can be applied for the analyses is the bottom zone except for the bottom surface area. Therefore, it is very important for the steelmakers to know the correct positions of a steel sample where inclusion determinations should be made for this particular lollipop-shaped sample. For example, the preferred positions that can be employed for inclusion determination are the bottom ½ centers and the bottom center.

For the study of the filling process of the rectangular-shaped sampler, the same physical modeling method and the same mathematical model are applied and discussed in *Supplement V*. Some difference can be drawn after comparing both the lollipop- and rectangular-shaped samplers:

1. The lollipop-shaped sampler is used in the industry, while the rectangular-shaped sampler is designed to be used during the laboratory trials;
2. The size of the rectangular-shaped sampler is much bigger than the other. Its filling time is in around 12s. While it is only about 1s for the lollipop-shaped sampler to be filled up;
3. The vortex region in the lollipop-shaped sampler is created in the initial filling and will enlarge in the end of the fillings. The vortex region in the rectangular-shaped sampler is also formed during the initial filling, but separated vortex regions (upper and lower) will formed in the end. They tend to close to the wall of the sampler;
4. Due to the various flow patterns in the different samplers, the inclusion concentrations, or density are different. More inclusions can be found in the center of the lollipop-shaped samples, while the inclusion density is very high on the surface of a rectangular-shaped sample.
Chapter 6

SUMMARY & CONCLUSIONS

The liquid steel sampling process of both the lollipop-shaped and rectangular-shaped sampler during its filling process has been investigated experimentally and numerically. In addition, the influence of initial solidification on the fluid flow and inclusion movements has been also studied. The most important specific conclusions from this study are summarized as follows:

i) Lollipop-shaped sampler:

1. During the sampling process, the flow pattern in the middle of the sampler vessel can be characterized into three distinct flow regions: 1) vortex flow region, 2) horizontal flow region and 3) the middle nozzle vertical flow region. A hump can be observed during the fillings;

2. For the calculation of flow field of the physical modeling, the Wilcox $k-\omega$ turbulence model provides the best predictions in comparison to the results obtained by the PIV experimental measurements. Specifically, the predicted vortex sizes and its center positions agree with the physical modeling results. Also, the vertical flow region in the middle of sampler above the inlet pin and the flow region beneath the vortexes both coincide with the PIV measurements;

3. The calculated flow pattern of molten steel in a production sampler was found to be very similar to that in the water vessel.

4. The solidification starts immediately after the liquid steel comes in contact with the metal mold. The rate of solidification is faster in the direction of sample thickness than the width direction. The sample surface at the bottom solidifies first followed by the upper surface and the inner body. Besides, the solidification can be influenced by the turbulent flow pattern during the filling;

5. The calculated turbulence energy dissipation rate in the sampler mold is very high compared to the value in the ladle during steelmakings. It also points out that the predicted inclusion collisions due to turbulence at sampling are much higher than those take place in a ladle furnace. The high collision volume is because of a high turbulence energy dissipation rate in the sampler. It is found that the collision volume in the sampler is about 60 times higher than that in the ladle furnace at the steelmakings;
6. The prediction of inclusion concentrations was carried out. The regions were investigated horizontally from the sample surface to center, and vertically from the top to bottom. The agreement of obtained tendencies was found to be good compared to the industrial plant trials on the sample surface. Besides, the concentrations of inclusions at other regions were also predicted. The calculated results show that the preferred positions that can be employed for inclusion determination are the bottom zone except bottom surface area. It estimates that the mean deviation between the calculated result and the predefined initial concentration for all inclusions in the bottom zone is within 10%.

ii) Rectangular-shaped sampler:

1. At the initial filling stage, the flow pattern is chaotic and disordered due to the geometry change from the inlet pin to the main sampler mold. It has been predicted that it may lead to a air involved which in turn, will cause the pore and porosity formation;

2. The employed Wilcox k-ω turbulence model well predicts the location of the vortex (recirculation flow) which is near the free surface region and close to the wall of mold. The results obtained agree well with the PIV measurement results;

3. Due to the recirculation flow close to the wall, higher inclusion density is predicted to present on the surface of a solidified steel sample. On the other hand, the inclusion density is very small in the center regions comparing with other regions and the initial value.
FUTURE WORK SUGGESTIONS

There are three suggestions for the future work concerning the numerical simulations:

1. The present numerical simulation is based on the Eulerian approach. Although reasonable results can be obtained, there are still difficulties to accurately analyze problems such as the shape of the gas/liquid interface changing continuously or the large deformation of the free surface. In this case, a particle method based on the Lagrangian approach can be employed to deal with such problems to accurately predict the filling process in the future. Some common methods can be used such as *Moving Particle Semi-implicit Method* (MPS), *Discrete Element Method* (DEM) and *Smooth Particle Hydrodynamics* (SPH);

2. Employ a more sophisticated solidification model instead of the simplified one. Solidification shrinkage, pores formation, segregation should be considered;

3. Also, the collision, break up and agglomeration of inclusion particles should be considered during the sampler filling process. The *Sommerfeld’s agglomeration model* can be applied for the inclusion simulations.
Nomenclature

Greek

$\alpha$ Collision Efficiency, Contact angle
$\Gamma$ Diffusion coefficient
$\delta$ Unit Tensor with components $\delta_{ij}$, Kronecker delta
$\varepsilon$ Turbulence Dissipation Rate
$\kappa$ Thermal conductivity
$\mu$ Viscosity
$\mu_e$ Effective Viscosity
$\mu_t$ Turbulence Viscosity
$\rho$ Density
$\sigma$ Surface tension
$\nu$ Kinematic Viscosity
$\phi$ Diameter, General fluid property
$\omega$ Specific Dissipation Rate
$D$ Diameter
$D_h$ Hydraulic Diameter
$F$ Volume fraction
$Fr$ Froude number
$g$ Gravity acceleration
$H$ Height, Melting heat
$H_L$ Depth of bath
$I$ Turbulence Intensity
$k$ Turbulence Kinetic Energy
$l$ Turbulence Length Scale
$L$ Length
$N_v$ Concentration of inclusions
$p$ Pressure
$Q_{L}, Q_w, Q_s$ Flow Rate
$r$ Inclusion radius
$Re$ Reynolds Number
$t, T$ Time, Temperature
$u$ x-velocity component
$u'$ Velocity fluctuations
$v, v, V$ y-velocity component, Velocity Vector, Volume
$w, W$ z-velocity component, With
$W_{ij}$ Collision Volume
$\nabla$ Vector Differential Operator

Roman

$A$ Surface area
$c_p$ Heat capacity
$d_{ai}$ Inner diameter
Reference

4. J. Yoshida, T. Ohmi and M. Iguchi: *ISIJ Int.*, 45 (2005), No.8, 1160
11. B. Li and F. Tsukihashi: *ISIJ Int.*, 45 (2005), No. 1, 30
12. L. Wang, Q. Zhang, C. Deng and Z. Li: *ISIJ Int.*, 45 (2005), No. 8, 1138
15. M. Söder, P. Jönsson, L. Jonsson: *Steel research Int.*, 75 (2004), No. 2, 128
Appendix

It is well known that the hydraulic diameter for a two-dimensional (2D) case, for example, considering a circular with a diameter of $D$, the expression for its area $A$ and perimeter $P$ are as follows:

\[ A = \frac{\pi}{4} D^2 \]  \hspace{1cm} (Eq.A-1)

\[ P = \pi D \]  \hspace{1cm} (Eq.A-2)

The ratio between equation (7) and (8) is:

\[ \frac{A}{P} = \frac{D}{4} \]  \hspace{1cm} (Eq.A-3)

Thus, the hydraulic diameter for a 2D case can be expressed as:

\[ D_h = 4 \frac{A}{P} \]  \hspace{1cm} (Eq.A-4)

Applying the same theory, for a three-dimensional (3D) case, we consider a spheroid with a diameter of $D$. The volume $V$ and total surface area $S$ can be expressed as:

\[ V = \frac{\pi}{6} D^3 \]  \hspace{1cm} (Eq.A-5)

\[ S = \pi D^2 \]  \hspace{1cm} (Eq.A-6)

The ratio between equation (11) and (12) yields:

\[ \frac{V}{S} = \frac{D}{6} \]  \hspace{1cm} (Eq.A-7)

Thus, the hydraulic diameter for a 3D case can be expressed as:

\[ D_h = 6 \frac{V}{S} \]  \hspace{1cm} (Eq. 2-5)