Physical and numerical modeling of multiphase flows during Continuous Casting

Per Engström
2014

Master of Science in Engineering Technology
Mechanical Engineering

Luleå University of Technology
Department of Engineering Sciences and Mathematics
To dad:

Hur underligt att veta:

du vet ej där du går,
att ljusa sånger spira
och susa i ditt spår.
Preface

When I now approach the finals of my mechanical engineering studies, I can almost not believe that I’ve studied for 5 years at Luleå University of Technology; it feels like yesterday I started my studies there. I’m glad that this Thesis is the last part of my studies; it feels like that now is the right time to start working.

I want to thank Ph. D. Pavel Ramirez Lopez for supervising me at swerea MEFOS, both in the project and in the English grammar. My examiner at the university Ph. D. Per Gren, thanks for all the support, work as well as questions about life. I would also like to thank Jonas Alexis for giving me this opportunity of doing my Master Thesis at swerea MEFOS. In addition, I would also like to thank all other employees at swerea MEFOS which helped me during different stages of the project.

I want to give a special thanks to Christer Olofsson at swerea MEFOS. Without you these measurements would not be possible. You’re often thinking like a researcher, but acts faster; you’re one of a kind. Also great thanks for all the laughs during the experiments, even though everything didn’t go exactly as planned.

Johan Svensson, Thank you for dragging me through school. Before, I couldn’t imagine finding someone that increased my studies so well. The study times have been great, even though the efficiency haven’t always been the highest. I really enjoyed all our discussions and problem solutions. It looks like our method of studying worked well for both of us.

A big thanks to all my family, without you, none of this had been possible. I am not even sure I would have ended as a mechanical engineer without pushes from Peter Ökvist. Without, I probably had gone out to sea. Future will tell if this was a better choice.

Per Engström
Luleå, June 13, 2014
Abstract

The quality of Continuous Casting (CC) products are highly connected to the flow of the liquid metal and injected argon inside the mould. Therefore, both physical and numerical modelling has been performed to investigate the metal flow inside the CC mould.

A physical model has been used for investigating the flow inside the mould. Therefore, it uses a re-circulating technique along with a Bi-Sn alloy for representing the liquid steel. Moreover, velocity measurements using a Vives electromagnetic probe have been carried out where typical CC flow structures such as jet, upper and lower rolls were found. Overall, good contact between the metal and the Vives probe was achieved.

The experiments showed that argon bubbles were possible to both count and classify using a high-speed camera with a framing rate of 500 frames per second. Almost 45% of the bubbles were discovered in the middle region of the mould (20% of the mould top surface area, central plane parallel to the wide faces). It was also found that the nozzle was misaligned.

Levels of both alloy and the covering silicone oil were measured simultaneously using a Light beam sensor, which measured both levels at high accuracy. The usage of argon made the oil dirty, which in time made impossible to measure the level using the Light beam sensor.

Argon bubbles along with a thin silicone oil layer were numerical modelled using a Discrete Phase Model combined with a Volume Of Fluid in a three dimensional model. This was solved using ANSYS Fluent 14.5.7. The fluid flow fields give streamlines that allows for further investigating of the flow patterns inside the model. In addition to these, more quantities can be post-processed for investigating different features of the flow. Also, bubbles were counted and it was shown that almost 50% of the bubbles were released within the middle region of the mould.
## Contents

1 Introduction ............................................. 1  
  1.1 Goals ............................................. 2  
  1.2 Continuous Casting Process ..................... 2  

2 Numerical model Theory .............................. 5  
  2.1 General Navier-Stoke’s equations ............... 6  
  2.2 Multiphase - Volume of Flow ................... 6  
  2.3 Argon injection - Discrete Phase Model ........ 7  

3 Experimental Setup and Measurement description... 11  
  3.1 Continuous Casting Simulator CCS-1 ............ 11  
  3.2 Velocity Measuring probe ....................... 16  
  3.3 Robot ........................................... 17  
  3.4 Measuring points for velocity ................. 17  
  3.5 Light beam sensor CHRocodile M4 ............. 18  
  3.6 High-speed camera ................................ 20  

4 Results and Discussion ............................... 23  
  4.1 Numerical Model ................................. 23  
    4.1.1 Two dimensional mesh and boundary conditions 23  
    4.1.2 Two dimensional post-processing ............ 25  
    4.1.3 Three dimensional mesh and boundary conditions 26  
    4.1.4 Three dimensional post-processing .......... 29  
  4.2 Continuous Casting Simulator CCS-1 ............ 36  
    4.2.1 Calibration measurements for the velocity probe 36  
    4.2.2 Velocity measurements .................... 42  
    4.2.3 Light Beam measurements ................... 48  
    4.2.4 Bubble detection from high-speed imaging . 50  

5 Conclusions ........................................... 57  

6 Future Work ........................................... 59  

7 Bibliography ......................................... 60
A  Vives reference measure plots  63

B  Unfiltered contour plots  67
1 Introduction

Today’s demands of high quality steel products at low costs necessitate studies of manufacturing processes such as Continuous Casting (CC). Overall in the world, various steel products are used for several different needs. The consumption and demand of quality steel are driving the industry to supply enhanced steel products. However, the quality of the final products are not depending only of the raw material used, they are also depending of the metal flow inside the process before solidification.

Liquid steel from blasting furnaces or electrical furnaces are intended to finalize as different products on the market, such as railway, beams, sheet etc. To enhance the product quality, steel plants first convert liquid steel into semi-finished products. One method to convert the liquid metal into semi-finished products is the CC-process [1], which is used at Scandinavian steel producers. The CC-process results in semi-finished products known as Slabs, which can be seen as lengths of steel with rectangular cross-sections. Poor quality of these slabs results in defects in the final products.

The CC-process involves the flow of hot liquid metal and interest lies on how it flows inside the mould. Due to the existence of both solidification (stress-strain behavior) and heat exchange, the flows can be quite of a challenge to study. The final quality of the cast products depends on several physical phenomena, which different defects originate from [1]. These defects make it challenging for the producers to attaining high quality products [2].

The flow inside the mould can be studied using either numerical modelling or physical industrial experiments. However, the liquid temperature over 1500°C [2] makes it both difficult and expensive to perform industrial measurements. Due to the rising computational power at lower costs, numerical modelling is a feasible alternative to study the phenomena inside the mould treating the multiple phases as well as heat exchange. As the numerical tools potential grows, future models will need to combine even more phenomena together [3]. In order to determine the quality of the numerical simulations and the used models, it is necessary to validate them through both industrial and laboratory
1.1 Goals

The goal of the project is to perform experiments in the MEFOS Continuous Casting Simulator, CCS-1 and perform multiphase simulations of the flow of this simulator in order to develop tools to compare and validate both. The final aim is to improve the techniques and understanding on how the metal flow behaves in CC-processes.

1.2 Continuous Casting Process

The main concept of the Continuous Casting process consists of hot liquid steel, which is poured into a cooled mould. The shell of the liquid metal begins to solidify and is then continuously withdrawn from the mould with the liquid metal inside it. In a secondary cooling process through water sprays, the liquid metal inside the shell solidifies.

The concept of Continuous Casting with an open water-cooled mould is shown in Figure 1. First, the hot liquid steel is transferred to the process in a ladle. The steel is then poured down to the tundish. This allows the flow from the tundish into the mould to be continuous even when the Ladle is changed. The tundish inside is covered with ceramic to withstand the heat from the liquid steel (around 1500°C). The liquid metal is transferred from the tundish to the mould using a Submerged Entry Nozzle (SEN) commonly made of ceramic material. The submersion of the nozzle prevents the liquid metal to oxidize and react with the atmosphere [2]. Thus, the design of the nozzle controls amount of metal delivered and the metal flow into the mould. It also controls the flow behavior of the metal inside the mould [3]. The mould side plates are water cooled which act as a primary cooling of the final semi-product, the cooling makes the outer surfaces of the metal to solidify and create a shell. The extracted metal is pulled by rolls where water mist is commonly sprayed on the metal as secondary cooling [2] and the extracted metal inside is cooled down. Finally it is either cut into slabs
as in Figure 1 or directly sent to the mill for rolling, known as strip casting [1].

Figure 1: Main concept of Continuous Casting with an open water-cooled mould [6].

Argon is injected through the SEN to prevent clogging and to carry inclusions of unwanted particles such as alumina up to the slag region [7]. The interaction between injection of argon (Ar) and the metal flow is important due to quality concerns. The main distribution of the argon bubbles shows up in the top region of the mould. However, previous models show that some bubbles are found in the lower part of the mould where they cluster together to inclusions and result in internal defects in the slabs. Some of the bubbles are also trapped into the meniscus region (the upper corner of the liquid steel), which make them follow in
the outer surface part of the final semi-finished products, causing surface defects [4]. These defects are known as "blowholes".
2 Numerical model Theory

The Continuous Casting process is a multiphase system with heat exchange. The metal flow can be numerically simulated using Computational Fluid Dynamics (CFD). The first numerical models for simulating the CC-process were introduced as early as 1970 by Brimacombe et al. [8]. These first models were focusing on heat transfer from the metal to the mould in one dimension (1-D). In the 1980’s, the first models for simulating the steel flow pattern inside the mould were introduced and were in two dimensions (2-D). When computers performance increased in the 1990’s, the flow simulations were in three dimensions (3-D). However, these models were not taking either phases or the meniscus region into account. Thus, some later studies had focus on the meniscus [2].

Until the late 90’s, there were no studies that coupled multiphysics problems such as CFD and solidification [2]. Modern computers have improved in performance and today it is possible to solve these types of problems in regular PCs. The computer evolution enables the size and the complexity of casting models to double every 1.5 years [3]. As described earlier, the CC-process involves both a multiphase system (liquid to solid of metal and slag in different phases) and heat exchange. In addition to these, CC-processes also involve argon (Ar) gas injection, which leads to even more computational efforts needed to perform these numerical simulations. Several previous studies treated the slag interaction interface as a no slip wall [4, 9]. However, this assumption results in faulty velocities near this interface. This resulted in difficulties in conclusions on how the argon bubbles travels near and within the slag interface; into the slag region.

The numerical simulation model used in this study, developed at swerea MEFOS [10] is based on the Navier-Stoke’s equations for compressible viscous flow. The interfaces between the different phases are tracked using the Volume Of Flow (VOF) method [2]. This approach has been validated earlier for different metallurgical flows [11]. Argon bubbles were introduced using Discrete Phase Model (DPM) [5]. The model used the RNG-\(k - \epsilon\) as turbulence model. In addition to this, some user define functions (UDF) were added to produce a realistic model.
theory about the used models are found in ANSYS Fluent 14.5.7 Theory Guide [12]. The fully model was solved using ANSYS Fluent 14.5.7 [13].

2.1 General Navier-Stoke’s equations

The Navier-Stoke’s equations generally referes to mass, momentum and energy conservation. The mass and momentum conservation equations are given by (1) and (2) respectively.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \tag{1}
\]

\[
\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \rho \vec{g} + \vec{F} \tag{2}
\]

The source term \( S_m \) refers to mass added to the continuous phase from dispersed second phase or any other user defined source. The static pressure is represented by \( p \) and \( \rho \vec{g} \) and \( \vec{F} \) are the gravitational body force and external body forces [12].

2.2 Multiphase - Volume of Flow

The VOF method uses a mixture density, \( \rho_{mix} \), and viscosity, \( \mu_{mix} \), are calculated as (3) respectively (4) [11];

\[
\rho_{mix} = \alpha_x \rho_x + (1 - \alpha_x) \rho_y, \tag{3}
\]

\[
\mu_{mix} = \alpha_x \mu_x + (1 - \alpha_x) \mu_y. \tag{4}
\]

Where \( \alpha \) represent the phase fraction for the different phases \( x \) and \( y \). Steel and slag can be represented by \( x \) and \( y \) respectively. This results that the continuity equation can be written as (5) [14]:

\[
\frac{\partial}{\partial t} (\alpha_x \rho_x) + \nabla \cdot (\alpha_x \rho_x \vec{v}) = \sum_{y=1}^{n} (\dot{m}_{yx} - \dot{m}_{xy}), \tag{5}
\]

where the overall velocity is described using \( \vec{v} \) and the mass transfer rate between the different phases \( x \) and \( y \) is described using \( \dot{m}_{xy} \) and
\( \dot{m}_{xy} \). Compared to the Euler-Euler method, VOF only solves one set of equations for both phases. The momentum equation using the VOF is written as \([6, 12]\).

\[
\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \left[ \mu \left( \nabla \vec{v} + \nabla \vec{v}^T \right) \right] + \rho \vec{g} + \vec{F} \tag{6}
\]

The VOF method produces a continuous transition from each phase. A transition between two phases will occur in the mesh interface between them. In this interface, the density and the viscosity will be a mixture of both phases. The VOF Method has been tested out to be an reliable approach for modelling metallurgical flows \([5, 10, 15, 16]\).

### 2.3 Argon injection - Discrete Phase Model

The argon bubbles are modelled in ANSYS Fluent 14.5.7 \([13]\) using the Discrete Phase Model (DPM) method, which uses the Euler-Lagrange approach. The Eulerian method refers to a non-moving reference frame and Lagrangian method to a moving reference frame. The DPM approach treats the fluid phase (liquid alloy) as a continuum and the bubbles are traced along the flow field. The trajectories are calculated by integrating the force balance on the bubbles (written in Lagrangian reference frame; locally for each bubble). The force balance can be written as \([7, 12]\):

\[
\frac{d\vec{u}_p}{dt} = F_D (\vec{u} - \vec{u}_p) + \vec{g} (\rho_p - \rho) + \vec{F}, \tag{7}
\]

\[
F_D = \frac{18 \mu C_D Re^2}{\rho_p d_p^2} \frac{24}{24}, \tag{8}
\]

where \( \vec{F} \) refers as additional acceleration, \( F_D (\vec{u} - \vec{u}_p) \) is the drag force per unit mass, \( \vec{u} \) is the fluid phase velocity, \( \vec{u}_p \) is the bubble velocity, \( \mu \) is the molecular viscosity of the fluid, \( \rho \) is the fluid density, \( \rho_p \) is the density of the bubble and \( d_p \) is the diameter of the bubble. The Reynolds number \( Re \) is defined as \([9]\):

\[
Re \equiv \frac{\rho d_p |\vec{u}_p - \vec{u}|}{\mu}. \tag{9}
\]
Collisions and coalescence between bubbles are included in the model. Collisions between two bubble originate from O’Rouke’s algorithm which assumes that two bubbles could only collide if they are in the same continuous-phase cell. In basics, the algorithm calculates the probability of the smaller droplet being within the collision volume. If the smaller bubble (index 2) is on a collision course with the larger (index 1), the center of the smaller bubble will pass within a flat circle centered around the larger bubble; the area of \( \pi (r_1 + r_2)^2 \), perpendicular to the smaller droplets trajectory. The collision volume is defined as this area multiplied with the distance which the smaller droplet has traveled \( v_{rel} \Delta t \). The probability of the collision is given by (10):

\[
P_1 = \frac{\pi (r_1 + r_2)^2 + v_{rel} \Delta t}{V},
\]

where \( V \) refers to the volume of the continuous-phase cell where the smaller droplet is known to be inside. This can be generalized for parcels, where there are \( n_1 \) and \( n_2 \) droplets in the larger respectively smaller parcel. The larger parcel will be affected with a mean expected number of collisions, \( \bar{n} \) as

\[
\bar{n} = \frac{n_2 \pi (r_1 + r_2)^2 v_{rel} \Delta t}{V},
\]

The probability distribution of the number of collisions is given by O’Rouke [12] and follows a Poisson distribution as

\[
P(n) = e^{-\frac{\pi \bar{n}}{n!}},
\]

where \( n \) refers to the number of collisions between the larger and the smaller bubble. For each timestep of DPM, the mean expected number of collisions is calculated for each pair of traced bubbles in each cell. A random sample from the Poisson distribution is generated to decide if or not each pair collides. If they collide head-on, the outcome is generally coalescence. Thus, the droplet bounces if the collision is more oblique. Bubbles can also produce turbulent eddies. The work done by the turbulent eddies on the bubbles is subtracted from the turbulent kinetic energy [17].
In addition, the bubbles are two-way coupled into the continuum phase; liquid alloy, for making it possible for the bubbles to interact with the liquid phase. This is achieved using momentum exchange and mass exchange. The momentum change when the bubble passes through each control volume is given by (13):

$$F = \sum \left( \frac{18\mu C_D Re}{\rho_p d_p^2} (u_p - u) + F_{other} \right) \dot{m}_p \Delta t, \tag{13}$$

where $\mu$ refers to viscosity of the fluid, $\rho_p$ density of the bubble, $d_p$ diameter of the bubble, $Re$ relative Reynolds number, $u_p$ velocity of the bubble, $u$ velocity of the fluid, $C_D$ drag coefficient, $\dot{m}_p$ mass flow rate of the bubbles, $\Delta t$ timestep and $F_{other}$ other interaction forces. In addition to this, the mass change is computed by the change in mass of a bubble as it passes through each control volume (14):

$$M = \frac{\Delta m_p}{m_{p,0}} \dot{m}_{p,0}. \tag{14}$$
3 Experimental Setup and Measurement description

Measurements with liquid steel in the industry are both expensive and difficult due to the temperature of the liquid metal (around 1500 °C [2]). Several experiments of water simulating the flow has been made in earlier studies, with state-of-art techniques such as Particle Image Velocimetry (PIV), Propeller flow meters and Laser Doppler Anemometry (LDA) for measuring velocities in water models [2, 4]. However, these models do not properly account the solidifying shell and interactions in the slag region at the meniscus region.

3.1 Continuous Casting Simulator CCS-1

MEFOS Continuous Casting Simulator (CCS-1) was designed and build between 2004 and 2007 during a Research Fund for Coal and Steel (RFCS) project. The CCS-1 simulates the continuous casting process including a tundish, stopper, SEN and a mould. CCS-1 is shown in Figure 3 along with a schematic view in [2, 5, 16]. The green arrows represent the re-circulation of the liquid metal. The hot liquid metal is transported from the mould (775x200 mm) up to the tundish and then back into the mould through the SEN. The flow rate of the liquid metal is controlled using a stopper, which can be moved up and down for increasing or decreasing the flow rate respectively. Note that CCS-1 is not intended to be used for studying the solidification process of the metal due to this design of re-circulation of the liquid metal.
Figure 2: Schematic view of CCS-1 and measurement equipment [5].
Figure 3: Continuous Casting Simulator CCS-1 [19].
The CCS-1 uses Bi-Sn alloy (58%Bi-42%Sn) for representing the metal of the Continuous Casting process. Since Bi-Sn alloy has a low melting point $T_{\text{melting}}$ of 137°C, it allows easier measurements and a safer work place for the experiments compared to a steel caster. CCS-1 specifications are listed in Table 1 and the Bi-Sn alloy properties compared to liquid steel in Table 2 [16]. The kinematic viscosity of Bi-Sn alloy is shown in Figure 4 as a function of temperature. The alloy was chosen considering its similarity with steel in flow properties and the alloy has also similar electrical properties, which makes it possible to test e.g. electro magnetic stirring. On top of the alloy, a silicon oil is used for simulating the liquid slag pool and to prevent the alloy from oxidizing due to contact with the surrounding air.

The casting simulator, CCS-1, can be used for a casting speed $v_c$ in an interval of $[0.6, 1.4]$ m/min. CCS-1 allows argon injection through the stopper tip at a flow rate in an interval of $[0, 14]$ litres/min. The casting

---

**Table 1: CCS-1 specifications** [16].

<table>
<thead>
<tr>
<th>Specification</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mould size</td>
<td>0.775 x 0.2 x 0.9 m</td>
</tr>
<tr>
<td>Tundish height</td>
<td>0.7 - 0.9 m</td>
</tr>
<tr>
<td>Argon flow rate</td>
<td>Variable: up to 14 lt/min</td>
</tr>
<tr>
<td>Immersion depth</td>
<td>Variable within 150 mm</td>
</tr>
</tbody>
</table>

**Table 2: Bi-Sn alloy properties compared to liquid steel** [16].

<table>
<thead>
<tr>
<th>Alloy Type</th>
<th>Viscosity, $\mu$ [$10^{-3}$ Pa·s]</th>
<th>Density, $\rho$ [kg/m³]</th>
<th>Kinematic viscosity, $\nu$ [$10^{-6}$ m²/s]</th>
<th>Electrical conductivity, $\sigma$ [$10^6$ 1/Ωm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel (1600°C)</td>
<td>6.3</td>
<td>7000</td>
<td>0.9</td>
<td>0.7</td>
</tr>
<tr>
<td>MCP 137 (150°C)</td>
<td>10.7</td>
<td>8580</td>
<td>1.25</td>
<td>1.0</td>
</tr>
<tr>
<td>MCP 137 (170°C)</td>
<td>8.6</td>
<td>8580</td>
<td>1</td>
<td>1.0</td>
</tr>
</tbody>
</table>
conditions used are shown in Table 3 and they were chosen for matching the numerical model.

![Figure 4: Kinematic viscosity of Bi-Sn alloy as a function of temperature [16].](image)

**Table 3: Casting conditions used in the experiment.**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Casting speed</td>
<td>0.75</td>
<td>m/min</td>
</tr>
<tr>
<td>Argon flow rate</td>
<td>4</td>
<td>l/min</td>
</tr>
</tbody>
</table>
3.2 Velocity Measuring probe

The high temperature of liquid metal makes it difficult to measure the flow field inside it. However, several probes to represent the flow field have been tested earlier in CCS-1; such as Vives probes and ultrasonic probes [5]. The Vives probes are designed by the principle that liquid metal distorts a surrounding magnetic field with a frequency proportional to its velocity [20]. The probe is constructed of a permanent magnet with a pair of electrodes as shown in Figure 5 [19]. The permanent magnet together with the electrodes induces a electromotive force which generates an electric field nearly proportional to the flow velocity. Earlier measurements of the CC flow field using Vives probes has shown that Vives probes are capable of determining the flow field inside the mould. The construction of used Vives probes made it possible to measure the velocity in the horizontal plane.

Figure 5: Schematic view of Vives probe [19].
3.3 Robot

A robot was used for moving the Vives velocity probe within the mould of the CCS-1, it is shown in Figure 6. The robot was controlled using a LabView script. The program made possible to configure the movement of the robot within a grid, defined separately for x, y and z with start coordinate, increment and total number of coordinates in each direction. All coordinates were referred in mm. All variables for each direction (x, y, z) could differ from each other. Above this, any coordinates could be added manually to the grid scheme. The coordinate system used by the robot has its origin in the top corner of the mould with the z-axis downwards, x-axis parallel to the mould narrow face and the y-axis parallel to the mould wide face.

![Figure 6: Robot used for moving the velocity probes](image)

3.4 Measuring points for velocity

The Vives velocity probe was controlled by the robot programmed with the grid shown in Figure 7, for each point (corner in the grid), the output
velocity from the Vives probe was sampled for 30 seconds at a sampling frequency of 5 Hz, which results in 150 samples for each point. The Light-beam was mounted at a distance of 215 mm from the SEN, measured out using a yard stick.

Figure 7: Measure grid for the Vives probe.

3.5 Light beam sensor CHRocodile M4

The metal level could be traced using a Light beam sensor shown in Figure 8 in the figure four probes (cylinders) are on top of the control unit (box). The Precitec CHRocodile M4 system consists of a main
control unit which handles the generation of white light. The light is then transferred to the probe inside a fiber cable. The probe then sends out the light and the results are transferred back to the control unit using the fiber cable. Results are sampled into a computer attached to the main control unit. The computer can be attached using USB, RS232 or RS422, depending on the user needs.

![Light beam sensor CHRocodile M4.](image)

Figure 8: Light beam sensor CHRocodile M4.

The light beam sensor is an optical distance measurement which can measure the position of a reflective or opaque surface. A schematic view of the sensor is shown in Figure 9. A Halogen lamp sends out white light onto the surface which is then reflected. Depending on the distance between the emitted position and the surface, different frequency of the color spectra is reflected. The method allows both fast and quite accurate measurements (±400 nm). Even with a glass cover for protecting the sensor, the sensor has provided reliable measurements earlier on CCS-1 [19]. It was possible to measure the thickness of oil covering the surface when the oil was relative transparent for white light.
3.6 High-speed camera

Argon bubbles are captured using a high-speed camera PHOTRON Fast-cam SA1.1, shown in Figure 10. The camera captured pictures with rate of 500 pictures per second. The camera was placed to view the top of the mould, to record the meniscus surface and the passage of bubbles through the surface shown in Figure 11. The placement figure is taken from the narrow face side of the mould. The objective used was Sigma 24-70mm F2.8 and it was placed at a distance of 700 mm above the mould top surface.
Figure 10: High-speed camera PHOTRON Fastcam SA1.1.

Figure 11: Placement of high-speed camera, taken from narrow side of the mould.
4 Results and Discussion

Numerical and physical modelling of CC has been carried out. The numerical modelling is presented first, followed by the physical modelling.

4.1 Numerical Model

The CC process were first analysed with a basic two dimensional case. Thereafter, a three dimensional case representing the experimental model was studied.

4.1.1 Two dimensional mesh and boundary conditions

A two dimensional model was earlier developed at swerea MEFOS to the CC process at SSAB. However, since previous results were not intended for comparison or validation. Thus, it differs from CCS-1 in dimensions. The model mesh is shown in Figure 12a. The mesh has a boundary layer close to the mould along with refinements at the meniscus level, within and near nozzle. The total mesh size is 60,839 hexahedral elements. Boundary Conditions are described in Figure 12b except the velocity inlets and pressure inlet, the edges are all no slip walls.
(a) Mesh of 2D CC mould.

(b) Boundary Conditions of 2D CC mould.

Figure 12: 2D CC mould model.
4.1.2 Two dimensional post-processing

At first, the model was solved for fluid flow using only basic flow equations with RNG (renormalization group) $k - \epsilon$ as turbulence model. When the solution had converged, the flow basics were analyzed. Contours of the velocity magnitude along with the streamlines of the two dimensional case is shown in Figure 13. Notice that only half is solved using symmetry conditions in the middle. The solution was then mirrored to represent a complete mould.

![contours and streamlines](image)

**Figure 13:** Velocity magnitude (m/s) in a two dimensional model of CC mould.

The argon bubbles were added after the initial flow was solved using two way coupled DPM model in Fluent. The model was solved for additional time to see the results of the DPM model usage; how the bubbles traveled within the molten steel and the resulting flow pattern in the alloy.
4.1.3 Three dimensional mesh and boundary conditions

A three dimensional model of CCS-1 was previously created at swerea MEFOS. The model is shown in Figure 14 with corresponding boundary conditions in Table 4. The mesh, Figure 15, consists of 3,016,158 elements, mainly hexahedra and tetrahedra elements, but some pyramids and polyhedra elements are also used. The metal level region was completely built up by hexahedra elements, as seen in Figure 15a, and were refined using Fluent refinement function. The silicon oil was patched as a thickness of 5 mm when the flow solution had converged. Surface tensions were added between the phases: air, alloy and silicone oil.

Table 4: Boundary conditions used in the CCS-1 3D model.

<table>
<thead>
<tr>
<th>Named Section</th>
<th>Notation</th>
<th>Boundary Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tundish walls</td>
<td>1</td>
<td>No slip wall</td>
</tr>
<tr>
<td>Stopper walls</td>
<td>2</td>
<td>No slip wall</td>
</tr>
<tr>
<td>Mould walls</td>
<td>3</td>
<td>No slip wall</td>
</tr>
<tr>
<td>Mould outlet</td>
<td>4</td>
<td>( v_y = 0.0934 \text{ m/s} )</td>
</tr>
<tr>
<td>SEN Walls</td>
<td>5</td>
<td>No slip wall</td>
</tr>
<tr>
<td>Tundish inlet</td>
<td>6</td>
<td>( v_y = 0.1624 \text{ m/s} )</td>
</tr>
<tr>
<td>Mould top</td>
<td>7</td>
<td>Pressure inlet</td>
</tr>
<tr>
<td>Tundish top</td>
<td>8</td>
<td>Pressure inlet</td>
</tr>
</tbody>
</table>
Figure 14: CAD model of CCS-1.
(a) Metal level region close up.

(b) Mesh in the symmetry plane.

Figure 15: Mesh of CCS-1 model.
4.1.4 Three dimensional post-processing

The three dimensional numerical model results of CCS-1 were post-processed using ANSYS CFDPost 14.5.7 [21]. Different quantities can be studied as velocities, pressure and bubble distribution (including sizes and coordinates). A 3D view of the model, including alloy surface, streamlines and bubbles is shown in Figure 16. The figure also includes a volume rendering of velocity magnitude.

Velocities in the liquid alloy are plotted as contours in the $y$ direction (Velocity $v$) in Figure 17, which represents the central plane ($x = 80$ mm). The velocity magnitudes are also plotted for four different planes in Figure 18. The numerical solution is solved with argon injections, which affects the angle of the jet flow. Different views with velocity contours are shown in Figure 19.

The top area was divided into three different areas to estimate the bubble distribution, as in the physical model (see Figure 36a). Each area sampled the bubbles passed using ANSYS Fluent built in function. The samples were then analyzed using MATLAB and the distribution of bubbles is plotted in Figure 20a and 20b. Even though it looks like a lot of bubbles passed through the outer regions, the flow in the middle region (20% region) is higher then the rest, which leads to that about 50% of the bubbles passed through the middle region, region 2. The bubbles along with their corresponding size are shown in Figure 21. It is seen from this figure that mainly small bubbles are traveling downwards in the mould. The major part of the larger bubbles is discovered near the nozzle while the smaller ones are more evenly distributed within the mould.
Figure 16: 3D view of CCS-1 numerical model.
Figure 17: Velocity contours in z-y plane at x = 80 mm (central plane).
Figure 18: Velocity magnitude contours in four different planes.
Figure 19: Numerical velocity magnitude results of CCS-1.
(a) Flow rate 4 l/min. Three different areas.

(b) Flow rate 4 l/min. Divided into two areas (Mid and rest).

Figure 20: Numerical modelled bubble distribution.
Figure 21: Bubble distribution in numerical model, front view.
4.2 Continuous Casting Simulator CCS-1

The experiments were carried out using the setup shown in Figure 22 with parameters casting speed \( v_c = 0.75 \text{ m/min} \) and argon flow rate 4 l/min.

![Fully setup used in CCS-1](image)

Before the test started, the amount of alloy in the mould of the CCS-1 were adjusted so that it would match the numerical model. Silicone oil was added for protecting the free surface of the alloy from oxidizing due to contact with surrounding air.

4.2.1 Calibration measurements for the velocity probe

The different Vives probes (460, 620, 771 mm in total length) were put into a calibration rig for measuring output voltage for different rotation speeds of the test rig. The calibration rig is shown in Figure 23. The circular disc was put into rotational motion using an electrical motor in
different rotational speed, 4 to 14 rpm. It was then possible to calculate
the tangential velocity $v_t$ for the different rotational speeds $n$ (given in
rpm) with known radius (0.22 m) of the calibration rig $r$ as given in (15):

$$v_t = \omega r, \quad (15)$$

$$\omega = \frac{2\pi n}{60}. \quad (16)$$

Least Square Approximation of the velocity $v$ as a function of voltage
$U$ along with each coefficient of determination $R^2$ given in (17) for the
620 mm Vives probe. An example of calibration data is plotted in Figure
24 for the 620 mm Vives probe. Minimum and maximum values for
200 samples for each rotational velocity $\omega$ are plotted together with the
Least Square Approximation for each probe and channel in Appendix A
(Figure 39, 40 and 41). As seen, the Least Square approximations are of
order 1: $v = C_1 \cdot U^1 + C_2 \cdot U^0$.

$$v(620\text{mm}, \text{ch0}) = 3709.0465 \cdot U - 0.019833, \quad (17a)$$

$$R^2(620\text{mm}, \text{ch0}) = 0.99737, \quad (17b)$$

A MATLAB script was created to investigate separate measurements
along with filtering results, the program GUI is shown in Figure 25. The
Signal Checker program showed that some of the calibration measure-
ments were started before the flow had converged due to change in rota-
tional velocity as shown in Figure 26. As exemplified in the figure, the
latest (newest) samples are used for calculating the mean value. In the
Signal Checker, this is illustrated by the gray area of the top plot, which
marks the values that are plotted in the bottom plot. The problem with
the non converged flow before sampling was solved by only using the
latest 200 samples from the calibration measurements.
Figure 23: Calibration rig.
Figure 24: Channel 0 of 620 mm Vives probe.
Figure 25: Signal Checker.
Figure 26: Non converged rotational speed of the liquid metal.
4.2.2 Velocity measurements

The Vives probe was then tested using argon flow rate of 4 lt/min. It was not possible to measure the velocity in the alloy using the Vives probes when argon injection was used. Therefore, the argon flow was turned off and the flow velocities in the alloy were measured. The measured grid with axis $x$ and $y$ shown in Figure 7 (p. 18) was used for the Vives probe. Only the middle plane ($x = 80$) was measured for the fully depth $z = [255, 585]$ mm. The other planes ($x = 15, 48, 112, 145$) were only measured to the level of $z = [255, 455]$ mm due to circumstances during the experiment. Measured voltages were filtered such that values above and below the standard deviation of the signal were ignored when calculating the mean values. The filtering was done to remove singular values which appeared when the probe became dirty (leading to poor electrical contact).

Contour plots of the velocity in $x$-direction (created using MATLAB contourf with interpolating using interp2) is shown in Figure 27 to 31. Above each contour plot, the top view of the mould is shown with the corresponding measure plane marked. Unfiltered values are to be found in Appendix B (p. 67). It is seen in the figures that the velocities are in the interval $[-0.1, 0.1]$ m/s. Notice that $x = 80$ (Figure 27) refers to the mid-plane. In this figure, one streamline is presented along with direction vectors for easier understanding of the contour figures. This streamline pattern is commonly referred as the upper roll. In the figure, a saturated area is also marked. This is treated to be an area with uncertainties due to the vibration of the probe when it reached the jet.
Figure 27: Velocity contour plot (m/s) of the middle plane at $x = 80$ mm.
Figure 28: Velocity contour plot (m/s) at $x = 15$ mm.
Figure 29: Velocity contour plot (m/s) at $x = 48$ mm.
Figure 30: Velocity contour plot (m/s) at $x = 112$ mm.
Figure 31: Velocity contour plot (m/s) at $x = 145$. 
4.2.3 Light Beam measurements

It was possible to measure both the metal level and the thickness of the oil in CCS-1. However, due to external circumstances, another case with different boundary conditions was used when the Light beam probe was in use. Nevertheless, these results show that the meniscus level could be traced along time together with the thickness of the silicone oil. It is important to know that the influence of argon bubbles make impossible to measure the thickness of the oil after some time since the oxides from the alloy accumulate into oil and makes it less transparent, with time black. Initially it was possible to measure the distance to the oil surface and the alloy surface underneath the silicone oil. In Figure 32 the measurement of the surfaces distances are shown. The lower curve represents the free alloy surface and the upper curve represents the oil surface level. The values outside the interval of ± one standard deviation of the signal were deleted to remove the singularity values. The measurements are shown in Figure 32. It is seen that the Light beam probe measures the thickness of the oil to around 3 mm. The levels are following each other very precisely within the measurement.
Figure 32: Light beam measurements.
4.2.4 Bubble detection from high-speed imaging

The high-speed camera view area is marked in Figure 33. The camera was mounted above the mould, pointing towards the mould top. This made it possible to record high-speed videos of the surface.

![High-speed camera view area](image)

*Figure 33: High-speed camera view area.*

The images should first be converted to 8-bit gray scale and then converted into a binary format. Afterwards, ImageJ could be used for detection of particles. This was tested using the Embryos example. The original picture along with the particle detection are shown in Figure 34.

The high-speed camera was not used with the same conditions as in the numerical model due to priorities on the experimental plan, the estimated casting speed $v_c$ used were 1.2 m/min. Nevertheless, the main idea was to estimate if it is possible to use the high-speed camera for
finding the bubbles and estimate their size and distribution.

Each video was recorded for approximately 10 seconds which leads to 5000 frames (500 frames / second) and were recorded for different argon flow rates. Different frames from the videos are shown in Figure 36 which represent the top view of half of the mould with the SEN to the left. The figures show a foam developed on the top of the alloy. The foam with its bubbles complicates the bubble estimation using mathematical models and software. The software \textit{ImageJ} \cite{18} was tested. The movies were converted to gray scale and then to binary. The function \textit{Analyze Particles} was then used without success. Some filtering of the image has to be done before counting of the bubbles. Therefore, the bubbles were counted visually to get an idea of the distribution. As seen in the figures of flow rate 5l/min (36b and 36c), the flow is moving the oil (dark) and open surfaces of the alloy are appearing (bright). This makes it even more difficult to estimate the bubble distribution.

The mould top view was divided into three different regions as in the numerical model (seen in Figure 36a), each region was then divided into three equal sized parts which made it easier to visually count the images. The bubbles were divided into three different sizes, described in Table 5.

The estimated eye counted distribution for argon flow rate of 5 and 7 l/min are shown in Figure 37 and 38 for the different area regions. It is
seen in the figures that the flow looks biased due to fact that the counted bubbles in region 2 and 3 are much larger amount than those counted in region 1. From the figures it is seen that around 40-50 percent of the bubbles are captured in the middle area region (central plane region) at 20% of the mould top area.

---

**Table 5: Bubble Sizes Estimation.**

<table>
<thead>
<tr>
<th>Name</th>
<th>Estimated Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>Small</td>
<td>&lt;5 mm</td>
</tr>
<tr>
<td>Medium</td>
<td>5-15 mm</td>
</tr>
<tr>
<td>Large</td>
<td>&gt;15 mm</td>
</tr>
</tbody>
</table>

**Figure 35: Argon flow rate 5 l/min with explanations.**
(a) Area regions.

(b) Argon: 5 l/min.

(c) Argon: 5 l/min, free surfaces.

(d) Argon flow rate: 7 l/min.

(e) Argon flow rate: 8 l/min.

Figure 36: High-speed images for different argon flow rates.
Figure 37: Visually counted distribution of argon bubbles at 5 l/min, $v_c = 1.2$ m/min.
Figure 38: Visually counted distribution of argon bubbles at 7 l/min, \( v_c = 1.2 \text{ m/min}. \)
5 Conclusions

Numerical and physical simulations of the Continuous Casting Simulator 1 (CCS-1) have been carried out. Both the numerical and the physical simulations of CCS-1 reproduces typical flow features known to be found in Continuous Casting; such as jet and rolls.

An automated procedure for calibration of probes and sequential measurements was designed to ensure the functionality of the Vives electromagnetic velocity probe. The Vives probe made it possible to measure the flow inside the physical model CCS-1. Inside the jet (at high velocities), the probe vibrates which affected the measurements. Outside the jet, the measurements were very stable and consistent. Along with the stable and consistent measurements, very good contact between the metal and the probe was achieved.

The High-speed camera was able to capture the bubbles at the oil-metal-interface at a rate of 500 frames per second. Bubbles were counted and classified, showing that almost 45 percent were released in the central area (20 percent of total area) and the rest towards the wide faces of the mould. Open surfaces along with bubble distributions showed that flow inside CCS-1 mould was somewhat biased and it was found that the nozzle was not aligned perfectly. Better illumination (maybe not AC powered) is needed to do a better estimation of the bubble size and their distribution.

The Light beam sensor showed that it was possible to perform point measurements on the mould level along with measuring the thickness of the silicone oil. Alloy levels were only possible to measure with clean oil; argon injection soils the oil with time which prevents the white light from the light beam sensor to pass through the oil.

The numerical model was able to predict the multiphase (air, alloy, oil) flow dynamics coupled with argon injection. Refinements of the mesh at the mould level allowed adding of a thin silicone oil phase using Volume Of Fluid (VOF) model for all the different phases. The model also included surface tension between each of the phases.

Argon bubbles were possible to model by coupling a Discrete Phase Model (DPM) along with all the other phases modelled with VOF. This
DPM model is describing a realistic distribution of bubbles within the mould using additional source terms. Bubbles were sampled in Fluent and post-processed using MATLAB which showed that the main part, 50% were released within the middle region. This was also observed from the high-speed imaging. Improvement of the numerical model is possible using inputs from experimental data. It was also discovered that the larger bubbles are found near the nozzle, while the smaller ones are more evenly spread in the mould.

The numerical model along with the experiments allows a deeper understanding of the mechanisms responsible for achieving stable flow. Stable flow leads to higher quality products during Continuous Casting.
6 Future Work

It had been of interest to measure all the velocity planes in CCS-1 in full depth, even deeper than these measurements. This would make it possible to fully find the lower roll etc. By analyzing the frequencies of the velocity within the jet, this frequency may be a function of the velocity. An accelerometer mounted where the probe-robot attachment would also be of interest. In addition, measurements of the velocity in z direction (parallel to SEN) would make the streamlines even better determined.

The coordinates from the robot could be mapped to Vives velocity measurement. This leads to continuous measurements of velocities and could also result in higher resolution.

Removing the oil together with better illumination could make it possible for estimating the argon bubbles computational. However, the alloy has to be covered with something to prevent it from oxidizing with the surrounding air.

The numerical model could be improved for better representation of the fluid inside the mould from results of CCS-1 experiments.
7 Bibliography


A  Vives reference measure plots

Minimum and maximum values for 200 samples for each rotational velocity $\omega$ are plotted together with the Least Square Approximation for each probe and channel in Figures 39, 40, and 41. Notice that the approximation equations are defined in Volt and the $x$-axis in figures is scaled in $\mu$V. To estimate the quality of an least square approximation, $U_{\text{approx}}$ which approximates the velocity $v$ as a function of measured voltage $U$ ($x$-axis), it is possible to calculate the coefficient of determination $R^2$. This variable indicates how good the approximation model is to predict the fitted values. It can be defined as the residual variance from the fitted model as (18):

$$R^2 = 1 - \frac{SS_{\text{resid}}}{SS_{\text{total}}},$$

where the $SS_{\text{resid}}$ are calculated as the sum of the difference between the measured voltage $U$ and the model prediction $U_{\text{approx}}$ (19). The total residuals $SS_{\text{total}}$ are calculated by the variance of the measured signal $U$ multiplied by the number of measurements minus one (20). The two equations can then be used together with (18) to calculate a final value of the coefficient of determination. The lowest value of $R^2$ in the approximation for the different probes are 0.992. This means that that model predicts 99.2% of the variance in the voltage $U$ which can be treated as good quality for the interval that voltage and velocity are measured.

$$SS_{\text{resid}} = \sum (U - U_{\text{approx}})^2,$$

$$SS_{\text{total}} = (#\text{measuring points} - 1) \cdot \text{variance}(U)$$
Figure 39: Reference measurements of 460 mm Vives probe.
Figure 40: Reference measurements of 620 mm Vives probe.

(a) Channel 0.

(b) Channel 1.
Figure 41: Reference measurements of 771 mm Vives probe.
B Unfiltered contour plots

The unfiltered versions of the contour plots presented in Section 4.2.2 are shown below in Figure 42 to 46. The SEN is seen in the right side of the figures and the mould and oil level are marked by a red double dashed line and a green line respectively. The values between the measurement points are interpolated using MATLAB.

![Contour plot](image)

*Figure 42: Contour plot of velocity (Vives) (m/s) with vectors of x = 15 mm.*
Figure 43: Unfiltered velocity contour plot (m/s) at $x = 48$ mm.
Figure 44: Unfiltered velocity contour plot (m/s) at $x = 80$ mm.
Figure 45: Unfiltered velocity contour plot (m/s) at z-y plane $x = 112$ mm.
Figure 46: Unfiltered velocity contour plot (m/s) at $x = 145$ mm.