Numerical modelling of centrifugal casting process

JUN YIN
Abstract

The centrifugal casting process is a common method for manufacturing the tubes, etc. Due to its high temperature and invisible mold, it is really difficult to know the mechanism of molten steel inside the mold. It is important to know the mechanism of the molten steel inside mold, since it will help the manufacturer to know more accuracy of the flow of the molten steel so that it can work for improving the productivity and quality of the products.

Casting funnel design is the designed by Akers for their funnel which will result in different flow behavior. In thesis work, casting funnel design will be investigated so that it can make sure that the casting funnel design can affect the flow behavior of molten steel or not.

Another method of changing the diameter of nozzle was also carried out and investigated with both simulation and experiment to changing flow behavior of molten steel. It will give Akers alternative method for changing the flow behavior to liquid steel.

The mechanism of solidification in centrifugal casting is also really important since it can give manufacturer the general view of solidification process. So solidification of centrifugal casting is also investigated in the thesis work.

**Keywords**: mathematical modeling, centrifugal casting, funnel, flow behavior, flow patterns.
Acknowledgments

First of all, I would like to express my sincere thanks to Professor Mikael Ersson for giving me a lot of guidance and suggestions through entire project period which enlighten my willingness for future research. Without your guidance and teaching, this work cannot be finished.

Secondly, I want to show my great appreciation to Dr Mats Söder for giving this opportunity to do my thesis work at Akers and your patients in explaining the centrifugal casting process at Akers also give me great help.

Special thanks the PhD student Yonggui Xu for his support to do the water model experiment and nice discussion with simulation setup. I would also want to thanks the PhD student Haitong Bai for his help to do the 3D printing of funnel.

I really enjoyed my time and learned a lot during my study in the department of Materials Science and Engineering especially in the unit of processes.
Contents

Introduction ........................................................................................................................... 1
Fluid flow .............................................................................................................................. 2
Rotational speed ................................................................................................................ 2
Åkers manufacturing process ............................................................................................ 3
Importance of centrifugal casting .................................................................................... 4
Literature review ............................................................................................................. 4
Aim and goal ..................................................................................................................... 5
Social and ethical issues .................................................................................................... 5
Numerical Model .............................................................................................................. 5
Basic fluid flow modeling ............................................................................................... 6
Basic governing equations ............................................................................................... 6
Conservation equation ...................................................................................................... 7
Transport Equations for the Realizable k-ε Model ............................................................ 7
Computational domain, mesh and boundary condition .................................................... 8
Boundary condition .......................................................................................................... 11
Experiment model ........................................................................................................... 11
Flow behavior of molten steel ......................................................................................... 15
Casting funnel design ....................................................................................................... 17
Solidification ..................................................................................................................... 18
Result and discussion ....................................................................................................... 20
Simulation of flow patterns of molten steel on the rotational wall ................................ 20
Casting funnel design ....................................................................................................... 24
2D axisymmetric model ................................................................................................... 33
Summary ............................................................................................................................ 41
Further work ..................................................................................................................... 42
Reference ........................................................................................................................... 43
**Introduction**

Centrifugal casting is a technique which is used for casting thin-wall cylinders. It uses the centripetal force to distribute the molten metal inside the mold and can cast materials such as: metal, concrete etc. The main difference between centrifugal casting and static casting technique are: the main purpose of centrifugal casting is to manufacture common material in standard size for further use, the static casting technique is forming the specialized shapes for current using. In centrifugal casting, there is a fixed mold which is rotating along the corresponding axis when the molten metal is poured from the ladle. The melts centrifugally move towards the mold wall because of the centripetal force. The melt will solidify after cooling. A schematic of the process can be seen in figure 1. The casting is usually a fine grain casting with a very fine-grained outer diameter, which is resistant to atmospheric corrosion, a typical situation with pipes. The inside diameter has more impurities and inclusions, which can be machined away [1]. Åkers is a company which is focus on producing the rolls.

![Figure 1 working principle of centrifugal casting [2]](image-url)
**Fluid flow**

The fluid flow is a really important factor in centrifugal casting process. With understanding of the fluid flow in centrifugal casting, it possible to improve the product quality and to produce the less defects. [3] In the manufacturing process at Åkers, it is very difficult to investigate the mechanisms of the fluid flow. The reason is that the system is opaque and at high temperature. Numerical simulation is a good and cheap method for studying and investigating the different controllable parameters which will influence the quality of final product. It is an alternative way to study and investigate the feature of fluid flow in the centrifugal casting process. When the molten steel coming into the system, it has higher viscosity, when it reaches the mold wall, the viscosity comes lower because of lower temperature. Then the molten steels start to solidify so that it can form the desired product. During this process, the melt cools and solidifies on the inner surface of the mold and simultaneously the melt, with enhanced viscosity, is picked up along with the mold wall to form a hollow cylinder. The feature involved during these processes is fluid flow [3]. It is obvious that the viscosity still plays a critical role in the centrifugal casting process. In this work, focus has been a parametric study of casting parameters such as nozzle geometry and casting speed.

**Rotational speed**

Another significant parameter in centrifugal casting process is the rotational speed. When pouring the molten steel from ladle to the mold, the friction between the mold and the molten metal result in rotational velocity being imparted to the molten metal. [4] With a high rotational speed, it will produce a high tensile stress product. A high rotational speed will result in high centripetal force (centripetal force is square proportion as rotational speed) which can create a strong convection in the liquid pool. With the low rotational speed, there are some problems to forming the defected product and produce poor casting. However, with the critical rotational speed, the uniform thickness of cylinder will be created inside the wall of mold. Van Heijst [5] has reported that when the fluid flow is beyond a certain rotational speed, the liquid metal will form a uniform thickness against the side wall. Thus, the rotational speed is a really important factor for the centrifugal casting process.
Åkers manufacturing process

Åkers is a company which has a long tradition to produce rolls. It has produced the rolls for over two hundred years. The Åkers group is not only the largest but also the oldest roll manufacturer in the world. [6]

The main equipment that Åkers use contains mold, funnel, ladle, nozzle etc. The first step is to produce the molten steel which is then stored in the ladle. The molten steel is than poured through a funnel. There are two thinner outlets which are connected with the funnel. The purpose of the tubes is to distribute the molten steel to the mold. During the process, the molten steel is transported by under the influence of a centripetal force. The centripetal force is acting on an axis toward the center of the axis. The equation is given in following Equation 1.

\[ F_c = \frac{mV^2}{r} = mrw^2 \]  

(1)

Where \( F_c \) is centripetal force\([N]\), m is the mass of molten steel \([kg]\), V is velocity\([m/s]\), r is radius\([m]\), and w is rotational speed\([rad/s]\).

Åkers also have their design of the funnel which called casting funnel design. The casting funnel design will be changed for different values of height so that it will change the inlet position in the mold. The main reason for this design is to try to change the flow of the molten steel in special situation. The casting funnel design will influence the flow of molten steel in the mold or not will be investigated in the project. It will be discussed more in the following paragraph.

The molten steel in the mold will have an even distribution due to the centripetal force. Solidification will also happen which will start on the mold wall. When the solidification is finished, the rolls have been roughly produced. The last process is to grind, coat, paint etc. It depends on the customer’s needs and requirements.

Casting funnel design

The aim of the design is to provide beneficial flow behavior of the molten steel. The method is to control the height of funnel so that it can reach desired requirement. Åkers also defines several values of the height. When those values change in the manufacturing process, Åkers wants to know how they affect the flow behavior of molten steel. Those values are got from Åkers.
Importance of centrifugal casting

There are several reasons why rolls are produced by centrifugal casting. First of all, the rolls which produce by centrifugal casting have good mechanical properties comparing with static castings. When molten steel pour into the rotational mold, the centrifugal force can help the molten steel to fulling the mold and the molten steel will have an even distribution in the mold. An even distribution of molten steel can result in less porosity and better mold filling. Besides, impurities such as dirt and sand slag can be removed. Due to those impurities are lighter, they can be easily collected on the inner surface of central hole. Furthermore, thermal gradients are much steeper because of unidirectional heat low in centrifugal casting system. The steep thermal gradient, especially with metal molds gives rapid solidification and therefore fine grain-size of products will be achieved.

Literature review

Kestur Sadashivaiah [3] has studied the effect of various variables on flow patterns. He carried out both numerical simulation and cold model experiment based on horizontal centrifugal casting. He also investigates the flow patterns based on different viscosities of liquid steel. R. Zagórski [7] presented a model and simulations of the centrifugal casting of a metal matrix composite reinforced with SiC in the initial stage of casting. Vinay Chandran [8] has studied the horizontal centrifugal casting process of manufacture aluminium-silicon carbide FGM by ANSYS fluent. He also compared his experiment result with the computational simulation result to verify the simulation result. J.W. GAO [9] has carried out a numerical simulation of solidification in centrifugal casting process of functionally graded materials. He found that the angular velocity, solidification rate and the geometrical nature of the particle flow are responsible for creation of the particle concentration gradient in solidified products. G. Chirita [10] presented that difference of production technology of a structure component with Al-Si alloys. The two production technologies involved were centrifugal casting and traditional gravity casting. The centrifugal casting technology gave a better mechanical property than the traditional gravity casting. S C Mondal [11] reported the process capability of the centrifugal casting process. He used the computational capability index value to analyze the results. He found that the Cu is not much variability in the index values. J Boháček [12] has studied the solidification process of work rolls which are made by the centrifugal casting process. A 2D numerical simulation was carried out to investigate the average flow dynamics. A water model was also set up for comparison and the results compared favorably with the numerical model for the transition to a developed flow regime. Based on the previous investigations above, the flow pattern in the vertical centrifugal
casting process is limited, so this project will give a mechanism on centrifugal casting process and it can also provide the valuable data on that.

**Aim and goal**

- Show the flow behavior of molten steel on the rotational wall in Åkers centrifugal casting system
- Show whether the casting funnel design can affect the flow behavior of molten steel or not.
- Show with different nozzle diameter, it can result in different flow behavior of molten steel or not.
- Show general view of solidification in Åkers centrifugal casting system.

**Social and ethical issues**

The aim of this work is to investigate the flow behavior of molten steel in Åkers centrifugal casting system. With this investigation, Åkers may improve their production which may lead to have good rolls products. The rolls are used in many areas such as factory, cars production, construction etc. With good properties of rolls, this area can make even better products as well. For example, car is common machine which is very useful for our society. People usually drive cars to work or travel. Thus, it can be said that this research work is also good for our society.

**Numerical Model**

In order to solve and investigate the flow behavior in Åker’s centrifugal casting process, finite volume method (FVM) has been used. The finite volume method (FVM) is nowadays widely used in industrial applications, including aeronautical, aerospace, automobile, naval and nuclear construction fields, and in applications of fluid mechanics, including tidal studies. [13] It
divided large problem into very small cells, simple equation and model that carry out in those small element cells will be assembled into large system that model the whole problem.

Computational fluid dynamics (CFD) using the FVM is applied in thesis work. CFD is closely based on the development of computer science and application of mathematic models. Comparing with experiment, CFD can give a visualized result which can help us understand the phenomenon. [14] But there are still some limitations for using CFD: storage and speed of computer, our inability to understand the mathematical model tec. Therefore, the more and more CFD commercial software have been widely used in fluid flow research.

### Basic fluid flow modeling

ANSYS FLUENT is computer software which is used for modeling the fluid flow, heat transfer and chemical reactions in complex geometries. It is equally suited for incompressible and compressible fluid-flow simulations [15].

Volume of fluid (VOF) proposed by Hirt and Nichols, was used to track the free surface [16]. By calculating a single set of momentum equations and tracking the volume fraction of each fluid through the domain, the VOF model can be used to model two or more immiscible fluid. The requirement for VOF model is that the two fluids are not interpenetrating. For air and molten steel, phases of air and molten steel are sharing a single set of momentum equation so that the volume fraction of each fluid in computational cells can be tracked through the domain.

The k-epsilon model is critical important in applied mathematical modeling of turbulence. It is a two equation model which means that it contains two equations to describe the turbulent properties of the flow. There are two transport variables which are turbulent energy k and turbulent dissipation $\varepsilon$. Besides, there are two major versions of k-epsilon model (the standard k-epsilon model and the realizable k-epsilon).

Solidification model is available in the Fluent CFD code which can be used for simulate the melting, solidification, casting, and crystal growth. This model is controlled by enthalpy of formation and it does not track the phase change during simulation.

### Basic governing equations
Conservation of mass (continuity equation)

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \]  

(2)

Where \( \rho \) is fluid density \([kg/m^3]\), \( t \) is the time \([s]\) and \( u \) is velocity vector.

Conservation of momentum

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

(3)

Where \( p \) is the static pressure \([pa]\), \( \bar{g} \) is the gravitational body force \([N]\) and \( \bar{F} \) is external body forces \([N]\).

Conservation of energy

\[ \frac{\partial}{\partial t} (\rho E_t) + \nabla \cdot (\rho u E_t) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (\Pi_{ij} \cdot u) + w_f + Q_H \]  

(4)

Where \( E_t \) is total energy per unit mass, \( \rho \) is density \([kg/m^3]\), the first term on the left-hand side of equation characterizes the rate of change of \( E_t \) in a control volume, while the second term on the left-hand side of the equation represents the rate of total energy transported by convection through the control surface. [17]

Transport Equations for the Realizable k-\( \varepsilon \) Model

The transport equation for k-\( \varepsilon \) model is:

\[ \frac{\partial}{\partial t} (pk) + \frac{\partial}{\partial x_j} (pk u_j) = \frac{\partial}{\partial x_j} \left( \left( u + \frac{u_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + G_k + G_b - p \varepsilon - Y_M + S_k \]  

(5)
\[
\frac{\partial}{\partial x_j} \left( p \varepsilon u_j \right) = \frac{\partial}{\partial x_j} \left( u + \frac{u_t}{\sigma_t} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left( \frac{\varepsilon}{k + \sqrt{\nu \varepsilon}} \right) + \frac{\varepsilon^2}{k} \gamma C_\varepsilon \varepsilon \right) + C_{1\varepsilon} \frac{\varepsilon}{k} \gamma C_{3\varepsilon} G_b + S_Z
\]  
(6)

Where

\[ C_1 = \max \left( 0.43 \frac{\eta}{\eta + 5} \right), \eta = S \frac{k}{\varepsilon}, S = \sqrt{2S_{ij} S_{ij}} \]

k: Turbulence kinetic energy
\varepsilon: Turbulence dissipation rate
u_t: Turbulence viscosity
\[ G_k \]: Generation of turbulence kinetic energy due to the mean velocity
\[ G_b \]: Generation of turbulence kinetic energy, because of buoyancy
\[ Y_M \]: Contribution of the fluctuation dilatation in compressible turbulence to the overall dissipation rate

C_2 and C_{1\varepsilon}: constant value which can be seen in Table 1.
\[ \sigma_k \] and \[ \sigma_\varepsilon \]: Prandtl number for k and \varepsilon, respectively as shown in Table 1

\[ S_k \] and \[ S_\varepsilon \]: User-defined source terms

<table>
<thead>
<tr>
<th>Table 1 constant value in Realizable k-\varepsilon Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_{1\varepsilon} )</td>
</tr>
<tr>
<td>1.44</td>
</tr>
</tbody>
</table>

**Computational domain, mesh and boundary condition**

The schematic domain, boundary condition and mesh are shown from figure 2 to figure 7. The geometric scale of the rotational mold and the funnel is applied from the steel industry [18] in millimeter. In order to save the simulation time, some air in the rotational mold was cut. In solidification simulation, the 2D symmetric model was used for saving the simulation time as well. In 2D symmetric model, it assumed that half of the molten steel was fulling the rotational mold where the red color act as the molten steel and the blue color represents air.
Figure 2 Schematic diagram of Åkers rotational mold

Figure 3 Cross section of computational mold

Figure 4 Bottom view of computational mold
Figure 5 Mesh of 2D symmetric model

Figure 6 Mesh of rotational mold

Figure 7 Phases of 2D symmetric model in the beginning
**Boundary condition**

**VOF and k-ε model**

Inlet boundary condition: the two circles in the top of the rotational mold act as inlet boundary condition.

Outlet boundary condition: outlet boundary condition is positioned at the top of the rotational mold which is pressure outlet boundary condition.

Wall boundary condition: the rotational wall boundary condition is used in the sidewall which has the special rotational speed. The bottom wall is defined as stationary wall which means that it works without any movement.

**For 2D symmetric model**

Rotational axis: the x-axis is defined as rotational axis which is used for symmetry.

Rotational wall: the rotational wall is the wall which is parallel with x-axis

Stationary wall: the bottom wall act as the stationary wall during the simulation.

**Experiment model**

In order to verify the simulation, the water model experiment was carried out at KTH water model lab. Due to the temperature effects were not considered since the impact of the temperature on the dynamic pressure of jets is relatively small. Besides, the Åkers centrifugal casting system is so big, it is difficult to test this mold in the water model lab. Thus, scaling of reduced mold was used in the experiment. The Modified Froude Number which is defined as the ratio of inertial force to buoyancy force was applied to make the model dynamically similar to the actual model.

\[
Fr' = \frac{\rho_1}{\rho_1 - \rho_g} \cdot \frac{V^2}{gH} \tag{7}
\]
Where $\text{Fr}'$ is the Modified Froude Number, $\rho_l$ and $\rho_g$ are the densities of water and air respectively $[\text{kg/m}^3]$, $v$ is the velocity $[\text{m/s}]$, $g$ is acceleration of gravity $[\text{m/s}^2]$, and $H$ is the characteristic length $[\text{m}]$.

$\text{Fr}'$ is constant value in the whole process. It was used to transfer the real model to the reduced one by using equation 7. In the thesis work, the model for Åkers was reduced by factor of 11 because of the size of the 3D printer. The detailed parameter information about water model shows is follow in table 2.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Model</th>
<th>Prototype</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometry</td>
<td>1:11</td>
<td>1</td>
</tr>
<tr>
<td>Number of inlet</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Density of material, kg/m$^3$</td>
<td>1000</td>
<td>7800</td>
</tr>
</tbody>
</table>

The 3D printer was used to produce the funnel. After the calculation of the reduced model, those values (from table 2) were used to the 3D printer for printing the funnel. There are also some geometry changes (different as Åker’s design) in the reduced funnel, since the real reduced funnel was difficult to setup in the experiment. The 3D printed funnel is appeared in figure 8, 9 and 10.

The main equipment is the container (in figure 11) which contains a hole in the bottom. The funnel needs to insert to the hole without any spacing between the hole wall and the funnel. This setup was used to make sure that the pressure was kept at a constant value during the whole experiment. The design of funnel with small circle in the bottom and big circle in the top can be
easier to achieve this requirement (it is shown in figure 8). Due to the diameter still remains the same value, this design has no influence to the final result. In order to make sure there is no spacing between the hole and the funnel, polyethylene adhesive tape was used and it is shown in figure 10.

Figure 11 Main container of water model experiment

During the experiment, the main container which is shown in figure 11 was fixed on the table without changing of location. One bucket was located under the funnel, it can avoid the water splash out. Water pipe sticks with adhesive tape on the wall of main container. The light bulb was located in the back and the high speed camera was putted in the front (see figure 12 and 13). All those setups have fixed location. The only thing that can be changed during the experiment is different diameter of the funnel.
When the experiment starts, the water was supplied by water pipe through the faucet. In the experiment, the height of the water was controlled with height 20cm so that it has the constant pressure at the bottom (20cm which was calculated by reduced model). The speed of water supply was controlled so that the height of water remains constant. When the height was constant and steady, the high speed camera starts to work.
Numerical model setup

In order to investigate the flow behavior of molten steel in centrifugal casting, the rotational mold was chosen as the simulation target in the beginning, since the whole system is very difficult to simulate and the most important part is the rotational mold. Thus, the system was divided into different parts: Funnel, mold, etc. The main work done in this project were: simulate the rotational mold to investigate the flow behavior on the rotational wall, use the different values for casting funnel design and compare them with simulation results, simulate the mold by using different diameter of nozzles, use the 2D symmetric model to simulate the solidification process.

Flow behavior of molten steel

In the beginning, the main work was trying the different simulation model to simulate the even simple mold. After the investigation, VOF and k-ε model with rotational wall were succeed with simple mold which can be seen in following figure 18.

![Figure 14 CFD simulation of centrifugal casting at 1,98 second](image)

Then the parameter data from the steel industry (Åkers) was used in the simulation. Due to the Åkers centrifugal casting system is really big; it took a lot of time to finish the simulation (almost 0.3s per day). In addition, the mesh was not good, since there are a lot of uneven cells. In order to improve the simulation result and decrease the simulation time, two inlet nozzles were deleted. Instead, the two circles in the top surface were used. The reason for this operation is to
optimize the mold meshing which can be seen from the following figure 19. Good mesh can result good simulation result and save the simulation time. After this operation, the meshing looks more uniform. However, it is still very slow, it can calculate only 0.4s per day. The main reason for this is that the Åkers’ system is too big. With the calculation, if the whole process wants to be simulated, it needs almost 250 days. It cannot be finished during this thesis work. So the simulation should be simplified. After discussed with the supervisor, the agreement has got that this project can just look the flow patterns when the molten steel just touch the bottom wall. So the whole mold system was carried out to simulate by using VOF and k-ε model and it costs two or three days to get the result. Compared with last simulation, this simulation (focus on the flow behavior of molten steel on the rotational wall) time decrease a lot.

In order to run it as fast as possible, some air in the rotational mold has been cut, since the air has almost no influence on the result (the flow behavior of molten steel on the rotational wall was focused). The method of cut material was used to achieve the target which can be seen in following figure 19 and figure 20.

![Figure 15 Cross section of rotational mold with meshing](image)
Casting funnel design

The schematic diagram of Åkers casting funnel design can be seen in figure 2. Åkers always control the height (in casting funnel design) so that the flow patterns and flow behavior of the liquid steel on the rotational wall can be different. Åkers want to know does this design can affect the flow behavior of molten steel on the rotational wall or not. The simulation was carried as the similar model with the previous one, since the different values of height will result in different location of inlet position. Due to the consideration of simulation time, three values were putted into the simulation (900, 885, 800). Compared with those simulations, it can be investigated some difference with those three values. The simulation of casting funnel design shows that there are some influences on the flow patterns and flow behavior of liquid steel. It will be shown in the part of result and conclusions. Base on the theoretical consideration, the design has been changed with diameter difference of inlet. Seeing following Equation 8.

\[ \dot{m} = \rho \times V \times A \]  

(8)

Where \( \dot{m} \) is mass flow rate \([\text{kg/s}]\), \( \rho \) is density \([\text{kg/m}^3]\), \( V \) is velocity \([\text{m/s}]\) and \( A \) is area \([\text{m}^2]\).

Due to the conservation, the mass flow rate should be kept at a constant value during the process. Density of molten steel has the same value all the time. Thus, velocity is only depending on area. Equation 8 can be transfer to following equation 9.

\[ A_1 \times V_1 = A_2 V_2 \]  

(9)
Where A is the area $[m^2]$ and $v$ is velocity $[\frac{m}{s}]$.

If the area of the inlet nozzle is changing, the velocity will be changed as well. Different velocity will result in different horizontal speed so that the position when molten steel jets reach the rotational wall can be different. Besides, liquid steel shapes will be influenced at the same time, since the diameter of nozzle was changed. Based on theoretical consideration, different diameter of nozzle will have more influence on flow behavior of steel.

Three value of diameter of inlet nozzle was simulated by using Ansys Fluent to have the test. In order to verify this design, the water model experiment has been involved to verify in KTH water model lab.

**Solidification**

In the beginning, the solidification model was added directly on the previous model of flow patterns on the wall. However, the result did not show anything about solidification. It shows only liquid phase in the simulation where no solid phase appeared. After theoretical study, it shows that the solidification process needs too much time to simulate. Thus, it cannot be seen on the rotational wall. Equation 10 is the energy equation used in Fluent for solving solidification and melting problem.

$$\frac{\partial}{\partial t} (\rho H) + \nabla \cdot (\rho \vec{v} H) = \nabla \cdot (k \nabla T) + S$$

Equation 10

Where $H=\text{enthalpy} [\frac{J}{mol}]$, $\rho= \text{density} [\frac{kg}{m^3}]$, $\vec{v} = \text{fluid velocity} [\frac{m}{s}]$, $S = \text{resource term}$

Figure 21 shows the principal example of copper which also has long solidification time.
As understanding that the solidification needs a lot of time to simulate, it cannot be finished during the master thesis work. After the discussion, the 2D symmetry simulation was carried out to figure out this problem, since the 2D symmetry simulation is much faster and can provide the general solidification mechanism in Åkers centrifugal casting system as well. The method carried out in the simulation is that it assumed the half of the liquid steel filling the rotational mold with temperature 1800K and then the system starts to rotate. The results will be shown in the part of result and discussion.

Figure 17 solidification of copper [19]
Result and discussion

Simulation of flow patterns of molten steel on the rotational wall

The simulation was carried out with rotational mold where the defined inlet velocity. It was calculated from the Equation 8 above.

VOF and k-ε model were used in the simulation setup. The numerical simulation was carried out by using commercial software FLUENT 17.0, the geometry and mesh were created in Ansys workbench. The detailed information about mesh qualities are summarized in the following table 3.

<table>
<thead>
<tr>
<th>Parameter and condition</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total number of Nodes</td>
<td>499982</td>
</tr>
<tr>
<td>Total number of elements</td>
<td>468027</td>
</tr>
<tr>
<td>Element quality (Min)</td>
<td>0.60007</td>
</tr>
<tr>
<td>Element quality (Max)</td>
<td>0.99999</td>
</tr>
</tbody>
</table>

In Åkers production process, the molten steel is poured directly to the rotational mold through inlet nozzles and it means that there contains some air in the rotational mold in the beginning. With continues pouring of the molten steel, the air will be decreased at the same time. In order to faster the simulation process, the air has been cut using cut material method, since the air has almost no influence on the flow behavior of molten steel. The boundary condition is defined in following table 4.

<table>
<thead>
<tr>
<th>Boundary condition</th>
<th>position</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet1 and inlet 2 outlet</td>
<td>Tow circle in the top surface</td>
</tr>
<tr>
<td>Rotational wall</td>
<td>Top surface with pressure outlet</td>
</tr>
<tr>
<td>Bottom wall</td>
<td>The sidewall which is rotate during production</td>
</tr>
<tr>
<td></td>
<td>Stationary wall in the bottom</td>
</tr>
</tbody>
</table>
The two circles in the top surface were defined with inlet velocity 3.18 in Y direction and -3.18 in z direction, -3.18 in Y direction and –3.18 in Z direction and the top surface is defined with pressure outlet as well. The bottom was defined with bottom wall which is stationary wall and side wall were defined with rational wall. The solution method used is coupled method.
Figure 19 Initial step of centrifugal casting process

Figure 20 Middle step of centrifugal casting process
Figure 19, 20 and 21 are the result from the simulation. It illustrates that the molten steel was injected from the two inlet nozzles with specific speed. It forms a small and straight molten steel jets which follow the direction of inlet molten steel. When liquid steel reach the rotational wall, there were some velocities from Y direction as well, since the rotational mold were rotated with specific speed which give the molten steel some speed at the same time. Due to the influence of rotational mold, the molten steel shape was distorted. The shape of molten steel was still cylinder, but it forms some small liquid particles in the later part due to drag force.
Casting funnel design

The three values that used in the simulation are 900mm 885mm 760mm. Where 900mm is biggest value, 760mm is smallest value and 885mm is used in present setup in Åkers centrifugal casting system. Due to the different values of height, the inlet position was changed. With geometry calculation, the inlet position was defined in following figure 11.

![Figure 22 Position of inlets](image)

The value of V2 is changing when height has different values.

**Table 5 V2 changing with height**

<table>
<thead>
<tr>
<th>Height</th>
<th>V2</th>
</tr>
</thead>
<tbody>
<tr>
<td>900</td>
<td>88</td>
</tr>
<tr>
<td>885</td>
<td>103</td>
</tr>
<tr>
<td>800</td>
<td>188</td>
</tr>
</tbody>
</table>
After theoretical consideration, with changing of height, only V2 is influenced. Based on the previous simulation of flow patterns on the wall, the inlet position was changed, other condition are remains the same. Following figure 23, 24, and 25 are the results.

Figure 23 flow behavior of Fasring 885

Figure 24 flow behavior of Fasring 800
From the figure above, it shows that the Åkers casting funnel design has some influence on flow behavior of liquid steel. In figure 23, the shape of liquid steel remains constant in the beginning and form some small droplets in the lower part. When height value changed to 900, the top part is almost same as height 885, but the lower part has some difference which contains some small liquid steel droplets. When the height values transfer to 800, there are more and big parties in the lower part. The formation of molten steel droplets may happen by the drag force. The molecules of molten steel which are close to the rotational wall have the same velocity with rotational wall. However, the molecules which are a little bit far from the rotational wall have lower velocity than the closed one. Thus, the drag force can be created so that the molten steel forms droplets in the lower part.

With the height changing, the position where the liquid steel reaches the wall was changed at the same time. In addition, the position where the molten steel reaches the bottom wall still has some difference. Thus, the result shows that the Åkers’ casting funnel design can influence the flow behavior of liquid steel.

With this design, it can result in some difference in lower part. But the top part where the liquid steel reach the rotational wall is almost same. Due to Åkers wants some smart design of funnel so that it can result different flow patterns. Parameter study of nozzle diameter follows the theory.
of mass flow rate was established. It obvious that mass flow rate has constant value in the whole process. Otherwise, the system cannot reach the balance.

With diameter difference, it can result in two parameters changing which are shape of the liquid steel and velocity. Comparing with height difference, diameter difference can result more parameter changing than height difference. In theory, it can have much influence on flow behavior of molten steel. So the simulation of diameter difference was carried out to investigate the flow behavior of molten steel.

**Parameter study of nozzle diameter**

The values that enter into the simulation are 30mm and 16mm. One is bigger than 26mm and one is smaller than 26mm, since it will be easier for comparison. The simulation setup is almost same as simulation of flow patterns on the wall. The only difference is the diameter of inlet. Based on the previous simulation model, this simulation was carried out to investigate the difference of flow behavior of molten steel. Results are shown in following figure 26, 27 and 28.
Figure 26 and figure 28 are almost same with molten steel flow behavior. But the figure 27 has much difference. First of all, the top part from inlet to the position where molten steel reach the
rotational wall becomes into the molten steel droplets. The reason for this is that high speed of molten steel with turbulence can create liquid droplets. Secondly, there contains much droplets in the lower part than the nozzle diameter 30mm and the nozzle diameter 26mm. Last but not least, the area of molten steel in the rotational wall becomes big which means that molten steel has much contact area with rotational wall. Comparing with figure 23, 24 and 25, it still the most powerful influenced on flow behavior.

With this simulation, it draws the conclusion that the smaller diameter of nozzle, the much droplets formation. Much particles formation will result much difference in flow behavior.

In order to verify this conclusion and simulation, the water model experiment has been setup for verification.

**Water model experiment**
Figure 29: Reduced funnel with diameter 2.2

- Uniform water jet
- Distance is about 3.9 cm
- Some small droplets
- Reduced with factor of 11

Figure 30: Self-designed funnel with diameter 26

- Uniform steel cylinder
- Some steel droplets
- Distance is 0.44 m
Figure 31 Reduced funnel with diameter 2.6

Figure 32 Self-designed funnel with diameter 30

Reduced with factor of 11

The distance is about 3.1

Uniform water cylinder

H₂

Uniform steel cylinder

Some small steel droplet

Distance is 0.36m
Figure 31 is the result from simulation with nozzle diameter of 26mm, the nozzle in physical model which has the diameter of 2.2mm, since the reduced model has been decreased with factor of 11. From the physical model, it can be seen that the uniform water cylinder was formed in the beginning. In the later part, there are some small water droplets formations. Comparing with simulation result, it has the same state. For figure 37, the molten steel droplets formation is located at lower part of liquid cylinder which is influenced by diameter of nozzle. Comparing with figure 34 and 36, the distance of the blue line is changed from 3.9cm to 3.1cm which is cause by different outlet velocity. The distance of blue line in physical model was created by two perpendicular line through the side of the funnel. When it reaches the water jets, the blue line was made for measurement. In computational model, the blue line was created by the height H (can be seen in figure 30 and figure 32). The H was then increased by factor of 11. Then the value was used in computational model to make the horizontal line which has the height of H11 (it is 11 times bigger than H). Because different diameter will result in different horizontal velocity so that the distance is different. The distance of the blue line is summarized in the following table 6 and table 7.

<table>
<thead>
<tr>
<th>Distance</th>
<th>Physical model</th>
<th>Multiply with factor 11</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nozzle diameter 2.2</td>
<td>3.9cm</td>
<td>0.429m</td>
</tr>
<tr>
<td>Nozzle diameter 2.6</td>
<td>3.1cm</td>
<td>0.341m</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Distance</th>
<th>Computational model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nozzle diameter 26</td>
<td>0.44m</td>
</tr>
<tr>
<td>Nozzle diameter 30</td>
<td>0.36m</td>
</tr>
</tbody>
</table>

The value from computational model is (0.44m and 0.36m) is much close to the one that calculated from measured distance (0.429m and 0.341m) during the experiment. The phenomenon from both simulation and experiment are almost same.
2D axisymmetric model

Due to the solidification process need too much time to finish the simulation, it has been explained in above part of numerical model setup. The 2D axisymmetric model was used to investigate the solidification mechanism. It was assumed that there was half molten steel with 1800k in the rotational mold. The model has been used are the VOF, k-ε, energy model and the solidification model. In order to make it rotate, the frame motion method was used. The results are shown from following figure 33 to figure 38. ( It uses the rotational speed of 9.2 rad/s )
Figure 39 Mass weight average of liquid steel

Table 8 Point information

<table>
<thead>
<tr>
<th>Time, s</th>
<th>Coordinate in x-axis, m</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.62s</td>
<td>1.9</td>
</tr>
<tr>
<td>74.8s</td>
<td>1.8</td>
</tr>
<tr>
<td>103.1s</td>
<td>1.7</td>
</tr>
</tbody>
</table>

From the figure 39, it shows that there formed “U” shape in the beginning. It is caused by centripetal force. Besides, there is a layer which mixes of both liquid and solid phase (because the layer is green color) above “U” shape, since the temperature outside is lower than the steel and energy transfer from lower temperature to higher temperature. From figure 39, 41 and 43, the layer of mixture was increasing with solidification time. The solidification happens much on the top part of rotational wall, since there are a lot of contact area with outside (contact with the rotational wall and air). It seems that the “U” shape height is decreasing during the solidification process. From table 10, it shows the information about point data where the height was decrease from 1.9 to 1.7 from time 3.62s to 103.1s. Therefore, it proves that the U shape height is reduced during the solidification process. With gradually decrease of “U” shape height, there is no “U” shape in the end. From figure 45, it illustrates the mass of liquid steel vary with flow time. It can be seen that the liquid steel drop very fast in the beginning and there is an sharp decrease of liquid steel from time 50s to 70s. Then the mass of liquid steel is decreased gradually which is more stable this time. The reason is that the mold was from stationary state to rotational state. When the model starts to simulate, the molten steel was transfer from a static state to movement
state. Thus, it is not stable and the “U” shape height is increased as well, since it is influenced more with centripetal force in the beginning. With the process goes deep, the system become to a stable one, so the mass of liquid steel decreased gradually.

The following results are also got from 2D symmetry model, but the value of rotational speed is about six times bigger than the previous one. This is also in real industrial cases. The results are shown from figure 40 to figure 47.

![Figure 40 Phases at time 0.1s](image1)

![Figure 41 solidification of mixed phases at time 94.8s](image2)
Figure 42 Solidification of mixed phases at time 173s

Figure 43 Solidification of mixed phases at time 232s
Figure 44 Solidification of mixed phases at time 275s

Figure 45 Phase at time 2.5s
Figure 46 Mass of liquid steel vary with time from 0s to 45s

Figure 47 Mass of liquid steel vary with time from 45s to 275s
Due to centripetal force, the molten steel forms “U” shape in the beginning which is shown in figure 40 (it is at time 0.2s). Then the entire molten steel stays on the rotational wall and rotates along the rotational axis. It is shown in figure 45. With the simulation time goes further, the molten steel still keeps in the same shape as shown in figure 45. It also shows in figure 46 that the mass of liquid steel keeps at the value about 6910kg. The reason is that the high centripetal force keeps the molten steel stays on the rotational wall. In order to speed up the simulation speed, only the energy equation is used in the later simulation. (the simulation with only energy equation was simulated from 45s). The molten steel starts to solidified from 94.8s which is shown in figure 41. Then the molten steel gets solidified on the rotational wall which is shown form figure 41 to figure 44. As close as the molten steel to the rotational wall, the deeper solidification it is. With investigation, it also shows that the molten steel on the top part of the rotational wall has deep solidification than the lower part. (it can be seen in figure 44). The reason is that half of the molten steel is assumed to fill the rotational mold with temperature 1800K which result in the temperature difference of top part of rotational wall and lower part. Comparing with lower part, the top part of the rotational wall has lower temperature.
Summary

- In Åkers centrifugal casting setup, the flow behavior is that the inlet nozzle has inject the molten steel into the rotational mold, then the molten steel goes straight to the rotational wall. After that, the molten steel flow from side wall to the bottom. It has almost uniform shape during the whole process (it forms some small droplets in the lower part of the mold).

- Gjuthuvudets placering (casting funnel design) design has the influence on the flow behavior. The most important part is that the lower part of the rotational mold has some molten steel droplets.

- With both simulation and theoretical consideration, it shows that the lower diameter of nozzle can result droplets formation in both the lower part and top part. The location where the molten steel reaches the rotational wall is still different. Besides, there are much more droplets in the lower part.

- Lower diameter of nozzle is most effective and powerful by influencing the flow behavior. It also gives Åkers an extra method to changing the flow behavior.

- Experiment in KTH water model lab validate the simulation of molten steel inject from nozzle.

- Solidification with 2D axisymmetric model provides the general view about solidification in Åkers centrifugal casting process.
Further work

- Validate the simulation of the molten steel behavior on the rotational wall.
- Take time to do the simulation of whole process (until the solidification complete)
Reference


[18] A. S. AB.

