CFD Simulation of Molten Steel Flow with Isothermal Condition in the Continuous Casting Tundish

Pruet Kowitwarangkul**, Maitri Kamonrattanapisud and Ekachai Juntasaro
Department of Mechanical and Process Engineering, The Sirindhorn International Thai-German Graduate School of Engineering, King Mongkut's University of Technology North Bangkok, Bangkok, Thailand

Deedeach Sukam
Bangkok Steel Industry Public Co., Ltd., Samuthprakarn, Thailand

* Corresponding author. E-mail: pruet.k.mme@tggs-bangkok.org DOI: 10.14416/j.ijast.2015.12.003
Received: 22 November 2015; Accepted: 17 December 2015; Published online: 25 January 2016

Abstract
One of the key factors to improve steel quality is the steel cleanliness. In the continuous casting process, the tundish serves not only as a reservoir and a distributor of molten steel, but also as a metallurgical reactor to diminish the inclusion content in the final product. This research is aimed to study the flow behavior inside the 3-strand tundish with and without flow control mechanisms. The commercial computational fluid dynamics software, ANSYS FLUENT, was used for simulation. The data of geometry and operating parameters were collected from the steel plant at Bangkok Steel Industry Public Co., Ltd., Samuthprakarn, Thailand. The simulations were performed under isothermal conditions. The results show that, with the implementation of flow control mechanisms, the residence time of the tundish flow was increased which would enhance the chance of inclusion removal and promote the steel cleanliness.

Keywords: Clean steel, Computational Fluid Dynamics, Tundish flow, Residence Time Distribution

1 Introduction

In modern steelmaking, steel cleanliness (or freedom from non-metallic inclusions) is a key factor in order to achieve a high quality steel product. Today, more than 90% of the world steel production is cast via tundish and continuous casting process. Tundish plays an important role to minimize non-metallic inclusions in order to improve the steel cleanliness.

Figure 1 shows a typical set-up of a continuous casting plant. In order to cast continuously, the molten steel from each ladle must flow and be reserved in the tundish and simultaneously distributed into each casting mold.

Although the tundish serves as a reservoir and

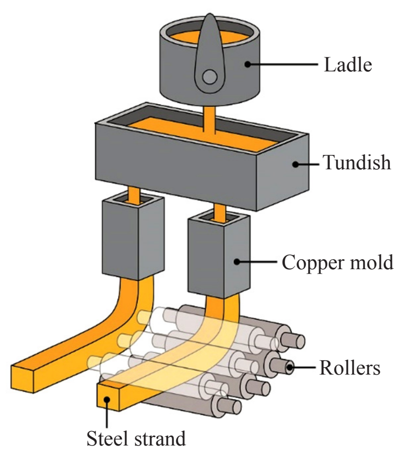


Figure 1: Diagram of continuous casting process [1].

Please cite this article as: P. Kowitwarangkul, M. Kamonrattanapisud, E. Juntasaro, and D. Sukam, "CFD simulation of molten steel flow with isothermal condition in the continuous casting tundish," *KMUTNB Int J Appl Sci Technol*, vol. 9, no. 2, pp. 71–77, Apr.–June 2016.

a distributor of molten steel, its function is also a metallurgical reactor. For example, the non-metallic inclusion, such as Al_2O_3 and SiO_2, can be floated and captured by the slag in the top layer of molten steel if there is enough residence time for the flow from inlet to outlet of tundish. Moreover, the homogeneity of the chemical composition and temperature in the molten steel also affects the inclusion content.

As the metallurgical performance of a tundish depends on fluid flow, the flow behavior in a tundish can be improved by incorporating suitable flow control mechanisms (i.e., Turbostop or pouring box, weirs, and other geometrical possibilities such as dams) and placing them at strategic locations.

The temperature of molten steel is so high, that almost none of adequate measurement technique is available for the investigation of flow behavior in the real tundish [2]. For this reason, the flow behavior is usually studied by using the physical water model and numerical simulations.

With the use of CFD (Computational Fluid Dynamics) simulations, the flow behavior can be visualized in several ways, e.g. the velocity flow field, the temperature field, the residence time of each streamline from inlet to outlet and the RTD (Residence Time Distribution). These data are useful for the design of tundish in order to diminish the inclusion content.

The articles [3]–[6] report the studies and research works about the tundish flow improvement in recent years. Regarding the tundish flow simulations, the standard or realizable k-ε model was mostly used for the turbulence modelling [3]. The results from [3] showed that, with the implementation of the flow control mechanism in the tundish, the streamlines of the molten steel had more opportunity to contact with the steel-slag interface at the top layer which promoted the inclusion seperation. Buĺko and Kijac [4] studied and modified the tundish working space by using the physical water model. They compared the residence time results of the non-equipped tundish and the tundish equipped with the flow control mechanisms such as Turbostop and weirs. Their results showed that the tundish modification prolonged the residence time.

Gastón *et al.* [5] did the numerical study and found that the difference between maximum and minimum temperature of molten steel in tundish was around 4°C. Such a small temperature difference justifies the use of an isothermal model in the current study.

Figure 2: Example of the 3-strand continuous casting tundish in the steelwork at BSI, Thailand.

In the current article, two types of tundish are choosen to study. Firstly, the 1-strand continuous casting tundish is numerically studied and compared with the experimental result from [6]. Secondly, the 3-strand continuous casting tundish from the steelwork at Bangkok Steel Industry Public Co., Ltd. (BSI), Samuthprakarn, Thailand as shown in Figure 2 is investigated and numerically modeled. The data of geometry and operating parameters are collected from the steelwork. The three-dimensional flow characteristics such as the RTD and the velocity field of the molten steel in the tundish with and without flow control mechanisms are simulated.

2 Methodology

In this study the numerical model of tundish flow is simulated by using the commercial software ANSYS FLUENT 14.0 which is one of the most recognised CFD programs. The flows in the current study are considered as turbulent flow, since the Reynolds numbers (Re) are equal to or greater than 5000. (The Reynolds number is calculate as $Re = (U*d_{hyd})/\nu$, where U is the cross-sectional average velocity, d_{hyd} is the hydraulic diameter of the tundish and ν is the kinematic viscosity). The numerical simulations of the turbulent flow are carried out on the basis of the Reynolds-Averaged Navier-Stokes (RANS) modelling.

2.1 *One-strand tundish model*

The CFD simulation of a one-strand tundish with the water flow inside is performed using the same parameters as used in the article [6] and is compared with the experiment results from the physical water model which is measured by Laser-Doppler Anemometry (LDA) as described in [6]. The aim is to validate the

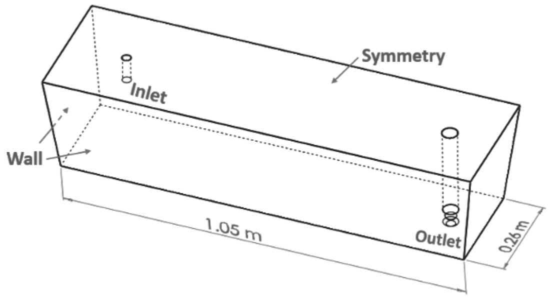


Figure 3: Geometry and boundary conditions of the 1-strand tundish model.

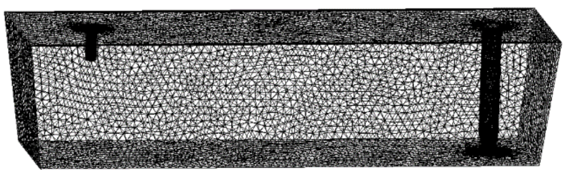


Figure 4: Mesh for the 1-strand tundish model.

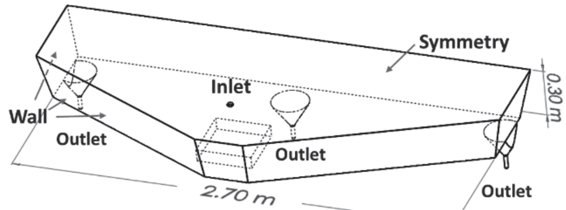


Figure 5: Geometry and boundary conditions of the 3-strand tundish model.

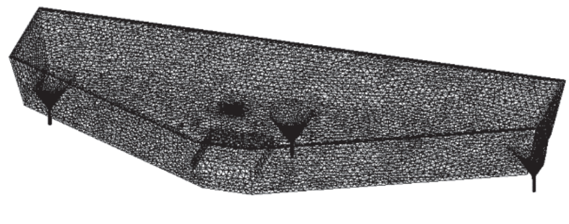


Figure 6: Mesh for the 3-strands tundish model.

simulation result of the current study. The tundish geometry and the boundary conditions are shown in Figure 3.

The top surface is assumed to be flat and is modeled by a symmetry boundary condition. An unstructured computational mesh of 0.47 million cells is used. The mesh is shown in Figure 4. The details of geometry, boundary conditions and fluid-property data are shown in Table 1.

Table 1: Detail of geometry and fluid-property data

		unit
Inlet diameter	0.023	m
Outlet diameter	0.023	m
Inlet velocity	2.4	m/s
Volume flow rate	1.0	dm^3/s
Turbulence intensity	3	%
Kinematic viscosity of water (20°C)	10^{-6}	m^2/s
Dynamic viscosity of water	0.001	kg/m·s
Water density	998.2	kg/m^3

2.2 *Three-strand tundish model*

The geometry of the 3-strand tundish model is created following the inner shape of the tundish sample from the steelwork at BSI. The geometry and boundary conditions of the model is shown in Figure 5. The top surface is modeled by the symmetry boundary condition. The walls and the bottom of the tundish are modeled by the stationary wall boundary condition. The molten steel flows through the inlet at the top and is modeled by the velocity-inlet boundary condition.

The outlets are set as the outflow boundary condition. An unstructured computational mesh of 0.5 million cells is used. The mesh is shown in Figure 6.

Table 2: Details of geometry and fluid-preperty data

		unit
Inlet diameter	0.03	m
Outlet diameter	0.014	m
Inlet velocity	3.0	m/s
Volume flow rate	2.12	dm^3/s
Turbulence intensity	5	%
Kinematic viscosity of molten steel (1556°C)	8×10^{-7}	m^2/s
Dynamic viscosity of molten steel	0.0067	kg/m·s
Liquid steel density at 1556°C	7000.5	kg/m^3

In this current study, the flow control mechanisms are designed and implemented into the 3-strand tundish model. Two dams are set up between 3 outlets and Turbostop (or the pouring box) is set up at the position

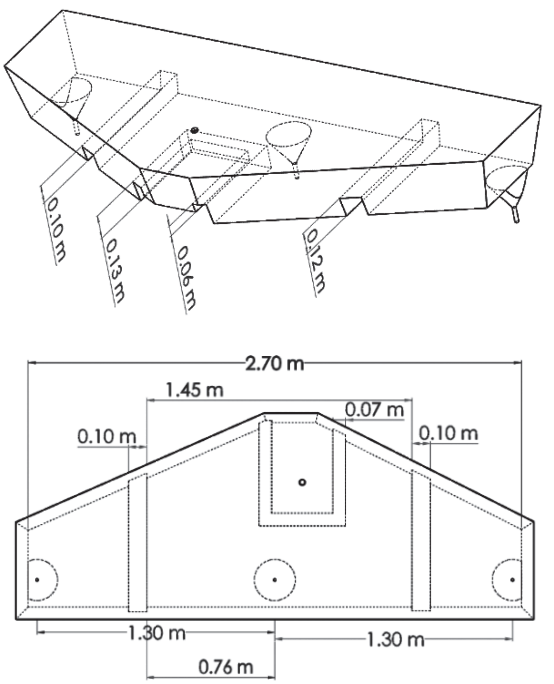


Figure 7: Geometry of the designed tundish with flow control mechanisms in 3-D view and top view The details of geometry, boundary conditions and parameters are shown in Table 2.

of the impact pad under the inlet as shown in Figure 7. The height of the tundish with flow control mechanisms is adjusted in order to have the same body volume as of the tundish without flow control mechanisms.

2.3 *Governing equations and methods used in ANSYS Fluent*

The simulations of tundish flow are performed by using ANSYS Fluent 14.0. The realizable k-ε model from Shih *et al.* [7] is used to simulate the turbulent flow. The SIMPLEC algorithm is used in the numerical simulation. The second order scheme is used to provide good accuracy. The simulations are performed under the following assumptions: (1) 3-D steady state, (2) Isothermal process.

The basic mathematical model equations describing the phenomena in the current study are as follows [8]:

$$\nabla(\rho u) = 0 \tag{1}$$

$$\nabla(\rho uu) = -\nabla p + \nabla(\bar{\bar{\tau}}_{eff}) + \rho g \tag{2}$$

$$\bar{\bar{\tau}}_{eff} = (\mu + \mu_t)[(\nabla u + \nabla u^T) - \frac{2}{3}\nabla u I] \tag{3}$$

$$\rho = 8300 - 0.7105\,T \tag{4}$$

where ρ is the density (kg m^{-3}), u is the flow velocity (m s^{-1}), g is the gravitational acceleration (m s^{-2}), T is the temperature (K), μ is the dynamic viscosity (kg m^{-1} s^{-1}), μ_t is the turbulent viscosity (kg m^{-1} s^{-1}), I is the unit tensor, $\bar{\bar{\tau}}_{eff}$ is the effective stress tensor (Pa), p is the pressure (Pa), Equation (1) is the mass conservation, Equation (2) is the momentum conservation, Equation (3) describes the effective stress tensor, and Equation (4) is the relationship between the density and the temperature of molten steel.

3 Results and Discussion

The CFD simulation in the current study is concerned with steady-state casting conditions. After finishing the calculation of tundish flow using ANSYS Fluent, the results are obtained and described as follows.

3.1 *One-strand tundish model: numerical vs experimental results*

The velocity flow field of tundish flow can be illustrated in a 2-D cross section plane at any interested position. Figure 8 shows the comparison between the velocity flow field from the numerical results and that from the experiment result of [6] at the cross section plane 0.2L as shown in Figure 8(a). Figure 8(b) shows the CFD simulation result from the current study and Figure 8(c) shows the experimental result from [6] of the water model using the LDA measurement.

The flow structure of the two counter-rotating toroidal vortices can be captured from the CFD simulation. The comparison of the flow pattern between the CFD simulation and the experiment shows good agreement with minor difference. The centers of the two counter-rotating vortices from CFD are a little bit lower than those from experiment. The average residence time of streamlines from inlet to outlet from the simulation in the current study is also calculated and is around 84 seconds, which also corresponds to the result from [6].

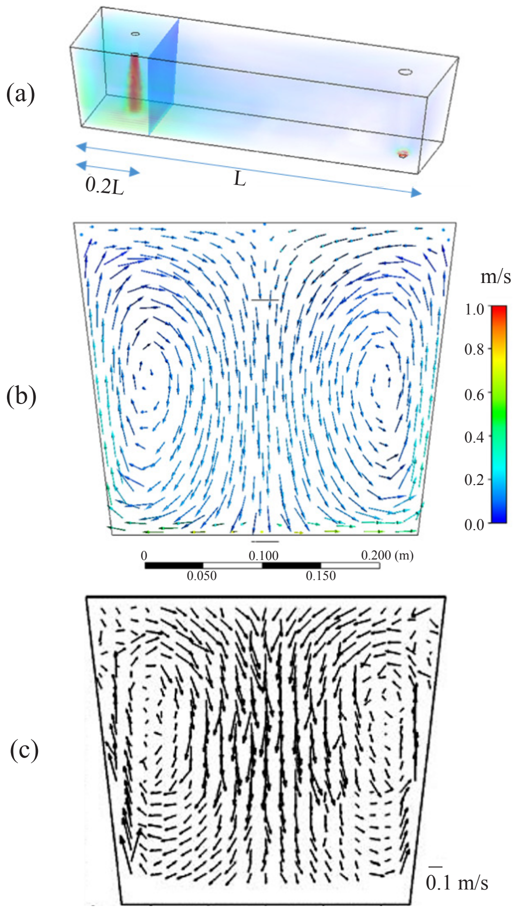


Figure 8: (a) Velocity flow field at the plane 0.2L
(b) Numerical result from the current study
(c) Experimental result from the article [6].

3.2 *Three-strand tundish model: the design of flow control mechanisms*

The flows of the three-strand tundish model with and without flow control mechanisms are calculated and converge after around 2000 iterations. The simulations are calculated under steady state and isothermal conditions at 1829 K. The CFD simulation results of the flow in the three-strand tundish models are described below.

3.2.1 Velocity flow field and streamlines

Figure 9 shows the comparison between the velocity flow field of tundish model with and without flow control mechanisms.

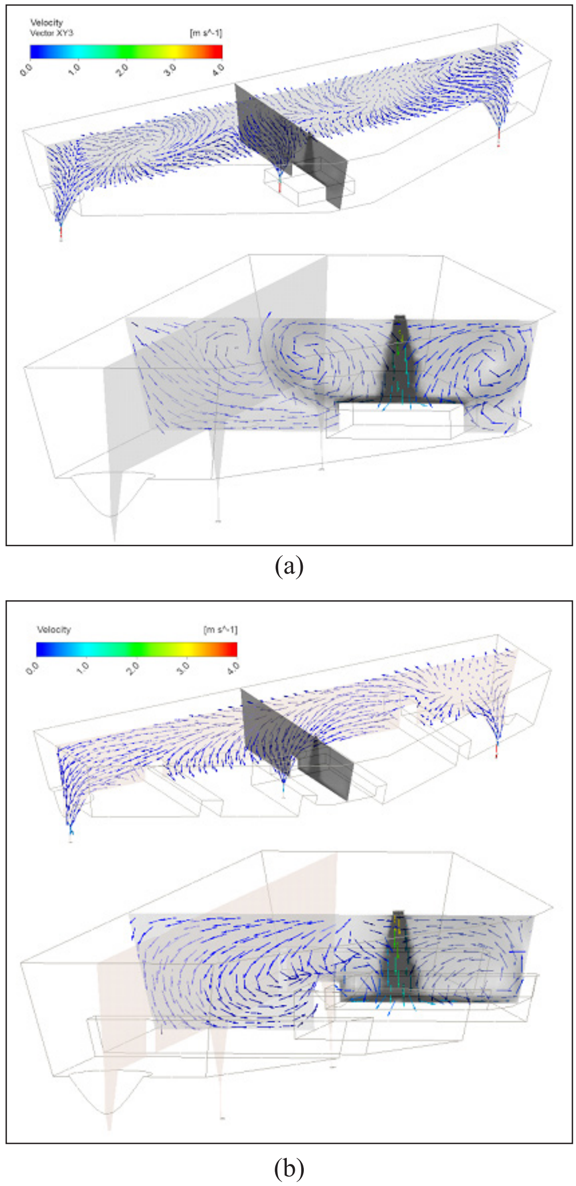
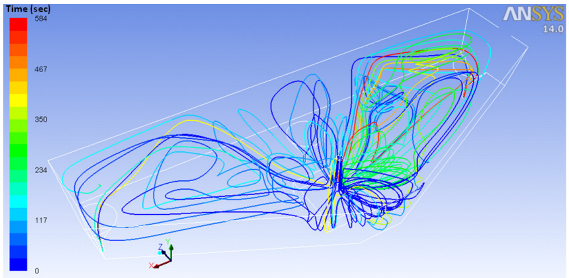
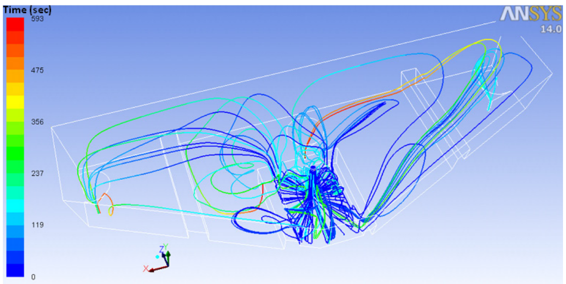


Figure 9: Velocity flow field in the three-strand tundish model (a) without flow control mechanisms (b) with flow control mechanisms.

The streamlines in both tundish models are created by the random injection of around 200 sampling particles which have the same density as of the molten steel. The streamlines of the outflow particles start from the inlet and escape the tundish at outlets. The examples of streamlines are shown in Figure 10.



(a)



(b)

Figure 10: Streamlines in the three-strand tundish model (a) without flow control mechanisms (b) with flow control mechnisms.

3.2.2 Residence time distribution

The residence time of each outflow particle in both tundish models is calculated from the inlet-to-outlet flow time. The residence time distribution in this current study is obtained by the arrangement of the residence time of each outflow particle into a group which has the difference of residence time within 50 seconds. The number of outflow particles in each group of residence time, e.g. a group of 0–50 seconds, is represented in percentage compared to the total number of outflow particles which has the residence time between 0–850 seconds as shown in Figure 11.

For example, in the tundish model without flow control mechanisms, the percentage of outflow particles with residence time in the range of 0–50 seconds is around 22% while, in case of the tundish model with flow control mechanism, the outflow particles is 0%. The peak of the residence time distribution also changes from the range of 0–50 to 100–200 seconds after the implementation of the flow control mechanisms into the tundish model. The comparison of the percentage of outflow particles in different residence time range

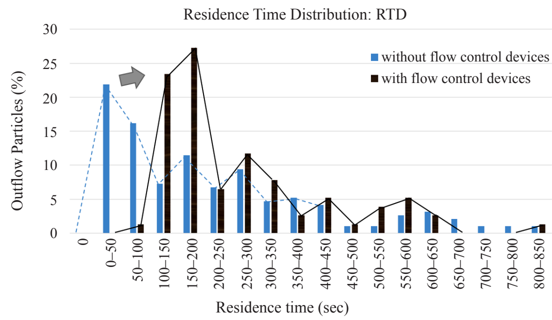


Figure 11: Inlet-to-outlet residence time distribution (RTD) of the flow in three-strand tundish model with and without flow control mechnisms.

and the average residence time of tundish model with and without flow control mechanisms are shown in Table 3.

Table 3: Average residence time and the percentage of outflow particles in different residence time range of tundish model with and without flow control mechanisms

	Without Flow Control Mechanisms	With Flow Control Mechanisms
Outflow particles with residence time of 0–50 seconds	22%	0%
Outflow particles with residence time of 100–150 seconds	7%	23%
Peak of residence time distribution graph	0–50 seconds	100–200 seconds
Average residence time	224 seconds	265 seconds

It can be seen that, after the implementation with the flow control mechanisms, the short-circuit streamline which has the residence time lower than 50 seconds in tundish model can be diminished, and the average of residence time of all outflow particles is also increased to around 20% as well. These will enhance the chance of inclusion to float and be captured by the slag at the top surface in tundish which will hence minimize the inclusion in the casting mold.

The average residence time of the tundish flow depends on several parameters such as the geometry, the volume, the flow rate and the flow control mechanisms. The 3-strand tundish model in the current study has the mass of around 4.6 ton and the average residence times are around 200–300 seconds. In case of the bigger tundish, e.g. with 22 ton capacity [9], the average residence time is as high as around 600–800 seconds. Therefore, the optimum average residence time is varying for each tundish. On the one hand, the prolongation of the average residence time and the minimization of the short-circuit flow enhance the chance of the inclusion floatation, but on the other hand, the too long residence time can be coincident with the dead zones or the stagnant zones. In tundish, dead volumes may occur owing to eddies trapped at sharp corners and might restrain inclusion removal in that area. The investigation of the dead zones in the current 3-strand tundish model is beyond the scope of this study and will be carried out in the future works.

4 Conclusions

From the current study, the one-strand and three-strand tundish models have been simulated using CFD program, ANSYS Fluent. The calculated results of the flow in one-strand tundish model such as velocity flow field and average residence time distribution show good agreement with the experimental results from literature.

Therefore, it is confirmed that numerical simulations are satisfactorily accurate. The three-strand tundish model is created based on the geometry of the steelwork in Thailand. With the implementation of the flow control mechanisms in the tundish model, the average residence time of the tundish flow from inlet to outlet is increased up to around 20%, which would enhance the inclusion removal and promote the steel cleanliness.

The results from the current study will support and enhance the potential for the steelwork to produce high quality steel in the future. It also saves the cost, instead of the design and testing in the real tundish process.

The future researches are such as follows: 1) to develop the physical water model in the laboratory scale in order to validate the numerical model of the tundish; 2) to investigate the dead zones or the stagnant zones in the tundish working space; 3) to develop the numerical model with non-isothermal condition; 4) to develop the numerical model for the trajectories of inclusions which have different density compared to the molten steel using discrete phase model, where the fluid phase of molten steel is treated as continuum and the dispersed phase of inclusions is solved by tracking a large number of particles through the calculated flow field.

Acknowledgments

This research was funded by "Coordinating Center for Thai Government Science and Technology Scholarship Students (CSTS), National Science and Technology Development Agency (NSTDA)" and "King Mongkut's University of Technology North Bangkok. Contract no. KMUTNB-NEW-59-13".

References

[1] N. Grundy, Continuous Casting: Optimizing Both Machine and Process with Simulation. [Online]. Available: http://www.comsol.com/story/continuous-casting-optimizing-both-machine-and-process-with-simulation-19247

[2] J. Huelstrung, "Development and Use of an Optimized SEN for the Thin Slab Casting Using Computational Fluid Dynamics," Ph.D. thesis, RWTH Aachen University, Germany 2006

[3] M. Warzecha. (2011). "Numerical Modelling of Non-metallic Inclusion Separation in a Continuous Casting Tundish," in *Computational Fluid Dynamics Technologies and Applications*, Prof. Igor Minin. [Online]. Available from: http://www.intechopen.com/books/computational-fluid-dynamicstechnologies-and-applications/numerical-modelling-of-non-metallic-inclusion-separation-in-a-continuouscasting-tundish

[4] B. Buĺko and J. Kijac, "Optimization of Tundish Equipment," *Acta Metallurgica Slovaca*, vol. 16, no. 2, pp. 76–83, 2010.

[5] A. Gastón, G. Sánchez Sarmiento, and J. S. Sylvestre Begnis, "Thermal Analysis of a Continuous Casting Tundish by an Integrated FEM Code," *Latin American Applied Research*, vol. 38, pp. 259–266, 2008.

[6] M. Warzecha, "Numerical and Physical Modelling of Steel Flow in a One-Strand Continuous Casting Tundish," *Metalurgija*, vol. 50, no. 3, pp. 147–150, 2011.

[7] T.-H. Shih, W.W. Liou, A. Shabbir, Z. Yang, and J. Zhu, "A new k-ε eddy viscosity model for high Reynolds number turbulent flows," *Computers and Fluids*, vol. 24, no. 3, pp. 227–238, 1995.

[8] *ANSYS FLUENT 14.0 Theory Guide*, SAS IP, Inc., 2011.

[9] T. Merder, J. Pieprzyca, and M. Warzecha, "Numerical Modeling of Steel Flow in the Six-Strand Tundish with Different Flow Control Devices," *Metalurgia*, vol. 48, no. 3, pp. 143–146, 2009.