Available online at www.sciencedirect.com

ScienceDirect

Materials Today: Proceedings 5 (2018) 9229–9237

materialstoday: PROCEEDINGS

www.materialstoday.com/proceedings

The 10th Thailand International Metallurgy Conference (The 10th TIMETC)

Flow prediction in the multi-strand continuous casting tundish of Millcon Steel PLC

Anawat Harnsihacacha[a], Adisorn Piyapaneekoon[a], Chanon Wattanaporn[b], Pruet Kowitwarangkul[a,]*

[a]*The Sirindhorn International Thai-German Graduate School of Engineering (TGGS), King Mongkut's University of Technology North Bangkok, 1518 Pracharat 1 Road,Wongsawang, Bangsue, Bangkok 10800, Thailand*
[b]*Millcon Steel Public Company Limited*

Abstract

The flow characteristics in the multi-strand continuous casting tundish of Millcon Steel PLC were investigated by CFD simulation and physical modelling. The aim of the study is to optimize the residence time distribution (RTD) which results in the better inclusion removal potential and the steel cleanliness. In this research, the 1:3 scale water model of the tundish was established. Tracer injections were computationally simulated using specie transport model and were physically simulated using red dye color mixed with sodium chloride. The flows are characterized by RTD curves which obtained from the tracer concentration measurement at the outlets. The results show that, among all outlets, the outlet in the middle position is critical and has very low of the minimum residence time. The flow modifiers can retard the minimum residence flow time and the mean residence time for all outlets which could promote the steel cleanliness.

Keywords: Multi-strand continuous casting tundish; Computational fluid dynamics; Physical water model; Flow characteristics

1. Introduction

With the increasing demand for the better quality of steel, steel cleanliness and inclusion removal play important roles in the continuous casting process. In traditional steelmaking, tundish acts as a distributor of molten steel

* Corresponding author. Tel.: +66 97 151 9070; fax: +66 2 555 2937.
E-mail address: pruet.k@tggs.kmutnb.ac.th

between the ladle and the mold as shown in Fig. 1 (a). Nowadays, tundish also acts as a metallurgical reactor in which the non-metallic inclusions such as Al_2O_3 and SiO_2 can have the chance to float and be captured by the slag at top surface of molten steel [1] - [3]. The opportunity of the inclusion floatation is related to the flow behavior inside tundish. If there is a lot of short circuit flow, where the molten steel flow from inlet to outlet in very short time, the chance of contaminated inclusion in the casting mold will be high. With the implementation of flow modifiers (such as dam and baffle) inside tundish, the residence time of the molten steel flow can be increased, which will enhance the inclusion removal potential.

CFD simulations and physical water model experiments are used to investigate the flow behavior and the inclusion removal performance of continuous casting tundish systems in several studies [3]. For the numerical study, the k-ε model is the most used model to simulate turbulent phenomenon of the flow in tundish [4]. Some researchers [5] found that applying the Reynolds stress model is somewhat superior to k-ε model, but it uses more CPU time. The study [6] and [7] show the examples of the tundish with flow modifier which prolong the residence time and could promote the inclusion removal potential. The results from the research [8] shows that even the inclusion particles with low diameter is able to flow to the top surface with proper tundish furniture design. In the study [9] a reduced-scale water model was developed and used to perform residence time distribution (RTD) experiments to determine the flow behavior. It shows that a bare tundish without flow modifier was proven to be insufficient in providing good flow properties for tundish operation.

The current research aims to study the flow behavior in the 4-strand tundish of Millcon Steel PLC with the condition that only 3 outlets are opened. CFD simulation and the physical water model were used to perform in this study. Two types of flow modifier, dam and baffle, were applied in the tundish model to improve the flow behavior. Fig. 1 (b) shows the picture of the 4-strand tundish at Millcon Steel PLC.

Fig. 1. (a) Diagram of continuous casting process. (b) The 4-strand tundish at Millcon Steel PLC, Thailand.

2. Physical water modeling and CFD simulation principles

2.1. Similarity criterion between the water model and the original tundish

In water model experiment, dynamic similarity and geometrical similarity between the model and the original tundish are required. For the dynamic similarity, Froude number in the water model (Fr_m) should be equivalent to that in the original prototype tundish (Fr_p) [2] as described in eq. 1:

$$Fr_m = \frac{U_m^2}{gL_m} = \frac{U_p^2}{gL_p} = Fr_p \tag{1}$$

, where U is the inlet flow velocity ($m{\cdot}s^{-1}$); g is the acceleration of gravity ($m{\cdot}s^{-2}$); L is the character dimension (m); subscript m is the water model parameter; subscript p is the original prototype tundish.

The geometrical similarity ratio of model to the original tundish (λ) in the current study is 1:3 as shown in eq. 2:

$$\lambda = \frac{L_m}{L_p} = \frac{1}{3} \tag{2}$$

The volumetric flow ratio of the model (Q_m) to the original tundish (Q_p) is

$$\frac{Q_m}{Q_p} = \frac{U_m \pi r_m^2}{U_p \pi r_p^2} = \sqrt{\lambda} \cdot \lambda^2 = \lambda^{\frac{5}{2}} \tag{3}$$

, where r is radius of inlet (m). From eq. 3, the flow rate of the experimental water model can be obtained, if the actual casting speed is known.

2.2. Numerical model

The numerical simulations of the tundish in the current study were carried out on the basis of the Reynolds-Averaged Navier-Stokes (RANS) modelling. The realizable k-ε model from Shih et al. [10] was used to simulate the turbulent flow. The basic continuity equations used to describe the fluid flow phenomena consist of the mass and momentum conservation equations [6].

2.3. Tracer injection simulation and RTD analysis

The flow characteristics of tundish were analyzed by the residence time distribution (RTD) curve. In numerical simulation of the current study, a tracer which has same property as of domain fluid was injected for 3 second into the tundish model on a steady-state flow field of the model and the concentration variation of the tracer with time is monitored at the outlet. Tracer injection was simulated using the species transport model with transient mode. For the flow characterization, the first step is to derive the dimensionless RTD-curve which shows the relationship between the dimensionless time (θ) and the dimensionless concentration (C_i) of strand i as described in [2], [7]. In the second step, the flow characteristics can also be classified by separating the flow volumes into two types: the active flow volume and the stagnant flow or dead volume (V_d). Active flow volume consists of the plug flow volume (V_p) and the well-mixed volume (V_m) as described in [2], [7].

3. Simulation and experimental setup

3.1. Tundish parameters and design of flow modifiers

In this study, the conditions of flow simulation in tundish was based on the information and parameters of continuous casting process at Millcon Steel PLC. The 1:3 scale tundish model compared to the original tundish was used in the study under the Froude similarity criterion. CFD simulations were performed and compared with the results from the physical water model. Two types of flow modifiers, dam and baffle, were designed and fitted in the tundish model as shown in Fig. 2. Table 1 shows the details of geometry and parameters of the original tundish at Millcon Steel and the 1:3 scale water model tundish.

The tundish at Millcon Steel PLC has totally 4 outlets, but it’s often the case that only 3 outlets are opened. So, in this study, the flow of 3 outlets in the 4-strand tundish model was conducted. The flow in tundish model was simulated by using the flow of water which has similar kinematic viscosity compared to the molten steel at high temperature. Froude similarity criterion was used to calculate the flow rate in the 1:3 scale tundish model in order to have the reliable depiction of the flow in the tundish model.

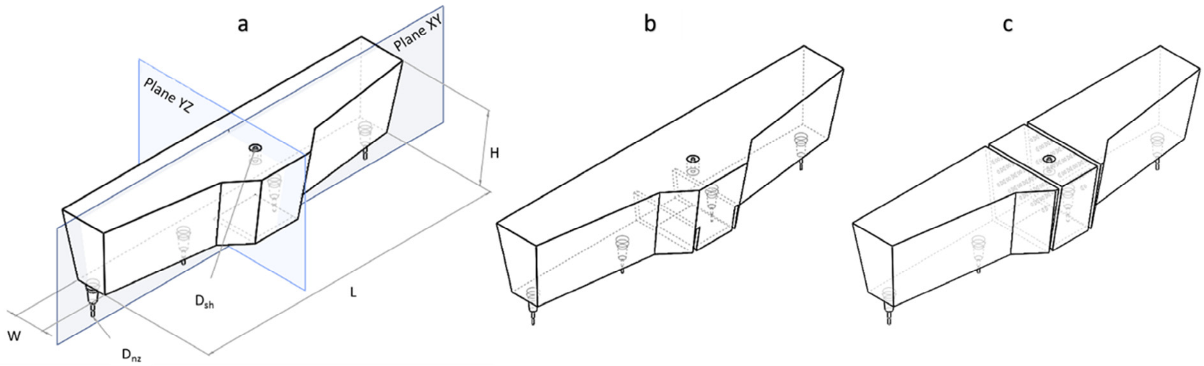


Fig. 2. Geometry of the tundish model with and without flow modifiers: (a) bare tundish, (b) tundish with dam and (c) tundish with baffle.

Table 1. Geometry and parameters of the original tundish at Millcon Steel and the 1:3 scale water model tundish.

	λ	Froude number	L	H	W	Shroud diameter (D_{sh})	Nozzle diameter (D_{nz})	Flow rate	Inlet velocity
Unit	-	-	m	m	m	m	m	(L/min)	(m/s)
Millcon Steel tundish	1	0.064	3.90	0.80	0.34	0.04	0.0155	128.25	1.7
Water model (1:3 scale)	0.333	0.064	1.24	0.27	0.12	0.012	0.0052	8.227	1.212

3.2. CFD simulation setup

In the current study the flow in tundish model was numerically simulated using the commercial software ANSYS FLUENT 14.0. The SIMPLEC algorithm was used in the numerical simulation. The second order scheme was used to provide good accuracy. The simulations were performed under the following assumptions: (1) isothermal condition, (2) 3-D steady state (3) transient during tracer injection and (4) the formation of waves at the free surface was ignored and the free surface was assumed to be flat and mobile.

Fig. 3 (a) shows the geometry and boundary conditions of the 4-strand tundish model with 3 opened outlets. Constant velocity, symmetry and constant pressure conditions were applied as the boundary conditions at inlet, top surface and outlet respectively. An unstructured computational mesh of around 7 - 12 million cells was used for the simulation. Two separated zones with different mesh resolution are shown in Fig.3 (b); the mesh in zone 1 near the inlet is finer than the mesh in in zone 2.

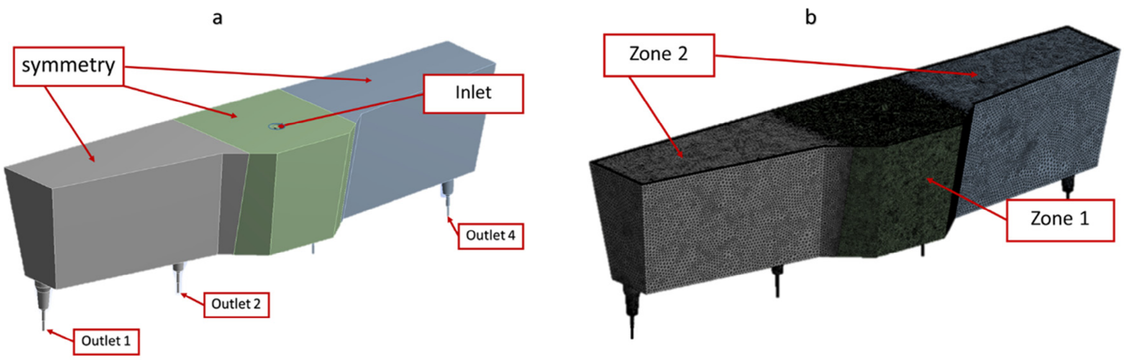


Fig. 3. (a) Geometry and boundary conditions of the 4-strand tundish model with 3 opened outlets; (b) Two zones with different mesh resolution.

3.3. Experiment water model setup

Fig. 4 (a) and (b) show the physical water model of the 4-strand tundish which is able to analyze the tracer injection and the residence time distribution. The height of water level was controlled to be constant at 27 cm. The flow at each outlet was measured by the flow sensor and controlled by the water pumps as shown in Fig. 4 (d). The system has a solenoid valve tracer injector combined with water feeding system. The solenoid valve system has two ways for feeding; the first one is for feed the water and another one use for switching to feed tracer into the tundish system. The tracer was mixed by 70 mL of water, 9 g of NaCl and 10 ml of red dye color. The tracer was kept in the small container as shown in Fig. 4 (c) and injected to the flow system by open the solenoid valve. The tracer concentration was measured at the outlet by the probe as shown in Fig. 4 (e) and recorded every 1 second by using electric conductivity meter.

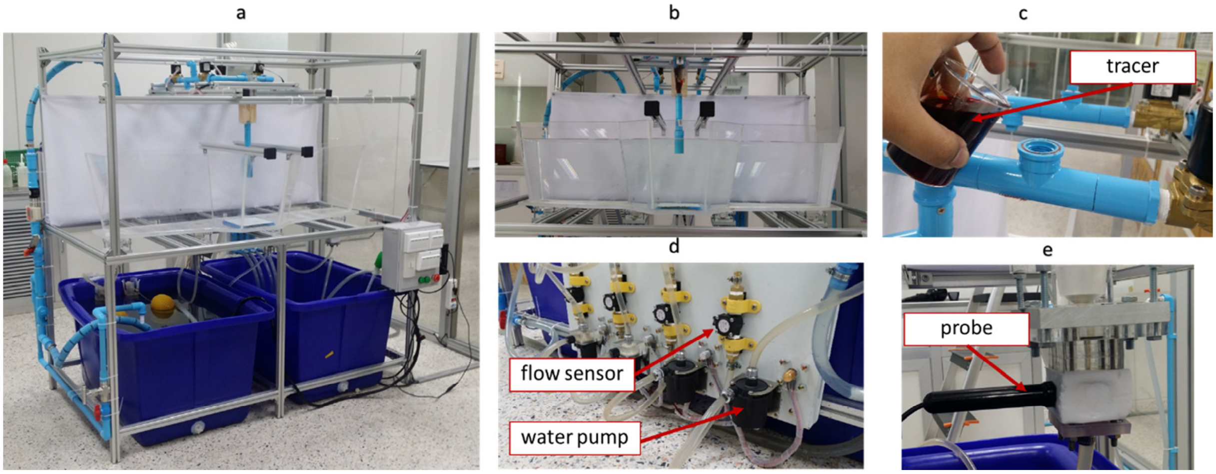


Fig. 4. Experimental setup of (a), (b) the physical water model of multi-strand tundish, (c) tracer, (d) flow sensor and water pump, (e) tracer measured probe.

4. Results and discussion

4.1. Velocity flow field

Velocity flow field in the bare tundish (tundish without flow modifier) and the tundish with flow modifiers such as dam and baffle are illustrated on the 2-D cross section plane YZ and plane XY as shown in Fig. 5. The arrow heads and color represent the direction and velocity of the flow.

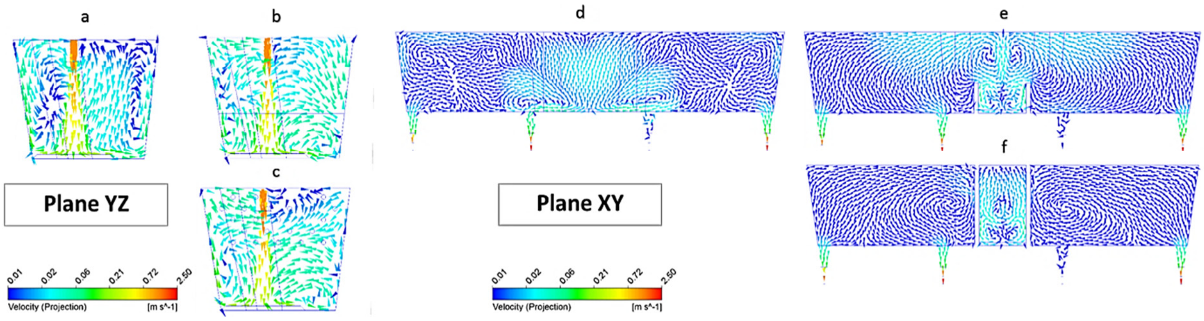


Fig. 5. Velocity flow field on the plane YZ of (a) bare tundish (b) tundish with dam and (c) tundish with baffle; Velocity flow field on the plane XY of (d) bare tundish (e) tundish with dam and (f) tundish with baffle.

The flow structure of the two counter-rotating toroidal vortices can be seen on the plane YZ of bare tundish as shown in Fig. 5 (a). The center of the left vortex has low velocity as shown by the blue arrow head. After fitting with flow modifiers likes dam and baffle as shown in Fig. 5 (b) and (c), the direction of the flow is changed and the velocity in the middle area is increased.

The velocity flow fields on the plane XY, which is the cross section plane that pass through the center of all outlets, are shown in Fig. 5 (d) – (f). In case of bare tundish, as shown in Fig. 5 (d), many vortices can be seen scatteringly on the plane XY and the green zone of high velocity flow covers wide area from the center to the zone of middle outlet-2. In this case, short circuit flows could take place from the inlet to the middle outlet-2. Flow modifiers like dam and baffle change the pattern of velocity flow field on the plane XY. It was found that after the dam installation, as shown in Fig. 5 (e), the area of vortex flow in the zone outside the dams is less than that of the bare tundish; the flow pattern in the center area of the plane XY is directed toward the top surface with slightly high velocity before swirling to the outlet direction. In case of the baffle installation, as shown in Fig. 5 (f), the area of the flow with low velocity on the plane XY (blue area) is wider than other 2 cases.

By comparison between 3 cases in Fig. 5 (d) – (f), it can be seen that the short-circuit flow from inlet to the middle outlet-2 could be reduced after fitting with dam and baffle.

4.2. Streamline

The streamlines in the tundish models are simulated by the random injection of 50 sampling particles which have the same properties as of the fluid. The streamlines of the tundish before and after fitting with flow modifiers are shown in Fig. 6.

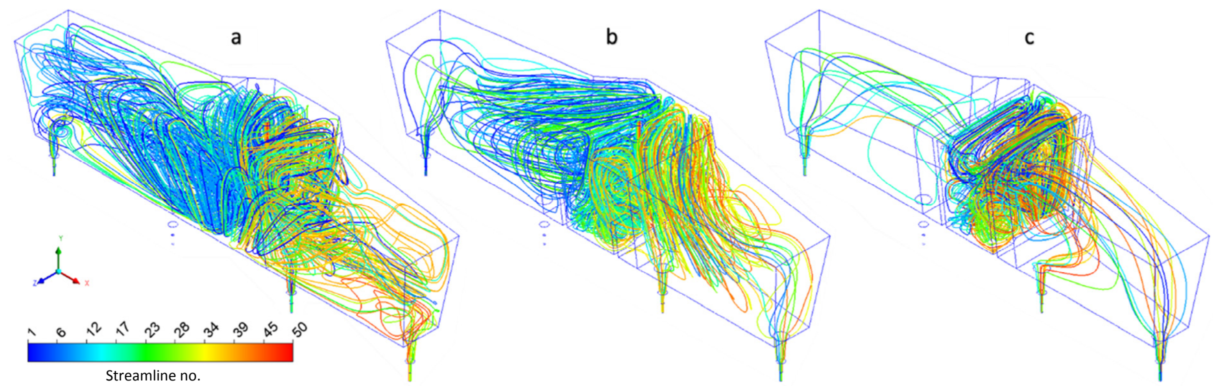


Fig. 6. Streamlines in (a) bare tundish, (b) tundish with dam and (c) tundish with baffle.

The color of streamline represent individual pathline of fluid flow from inlet to outlet. One route is represented by one color. Flow structure of fluid in bare tundish with many vortices can be seen scatteringly as shown in Fig. 6 (a). After fitting the tundish with dam or baffle, flow pattern was changed. In tundish with dam, as shown in Fig. 6 (b), many vortices occur at the zone between the dams and this flow pattern gradually disappear when it flows toward the outlet zone. In tundish with baffle, as shown in Fig. 6 (c), very high intensity of the vortices occur at the inlet zone between both baffles. After the liquid flow through the baffles, the intensity of the vortices was decreased. It can also be seen from this results that the short-circuit pathlines from the inlet to the middle outlet-2 are prevented after fitting with dam and baffle.

4.3. Tracer injection

The comparisons between CFD simulation results and visual observation of red dye tracer mixing phenomena in the water model for the bare tundish, the tundish with dam and the tundish with baffle are shown in Fig. 7. Tracer mass fractions in the fluid at 1 and 10 seconds after injection are shown with red-black color scale bar in Fig 7 (a) and (b) respectively. The CFD simulation results are in agreement with the experiment results. In case of the bare tundish, the incoming tracer from the inlet quickly enters the middle which represents the short circuiting flow. The use of flow modifiers, especially baffle, changes the flow patterns significantly and is able to prevent the short circuit phenomena.

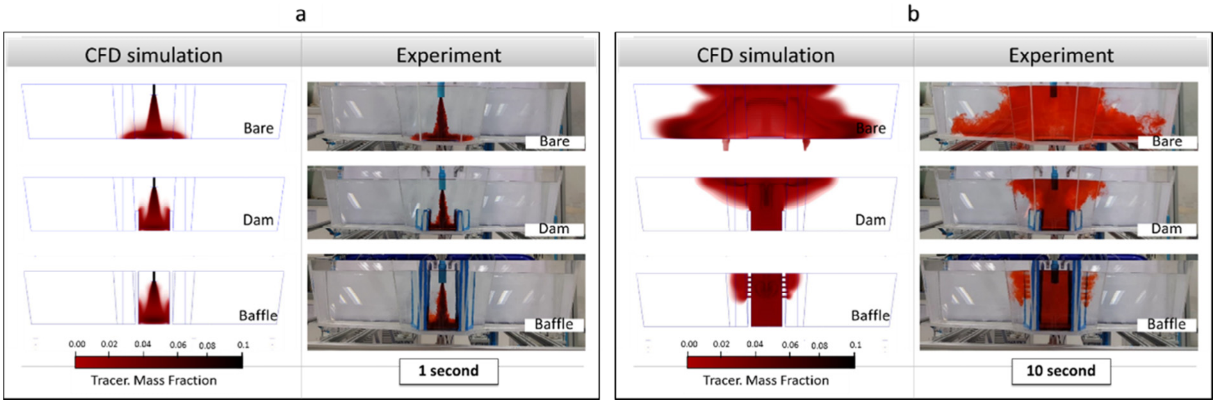


Fig. 7. CFD simulation and experiment results of tracer flow after injection for (a) 1 second and (b) 10 seconds.

4.4. RTD analysis

The residence time distribution (RTD) curves which show the relationship between the dimensionless concentration (C) of the tracer against the dimensionless time (θ) from 3 outlets of the tundish models with and without flow modifiers are represented in Fig. 8. Fig. 8 (a) – (c) shows the results from the CFD simulations. Fig. 8 (d) – (f) shows the results from the physical water model experiments.

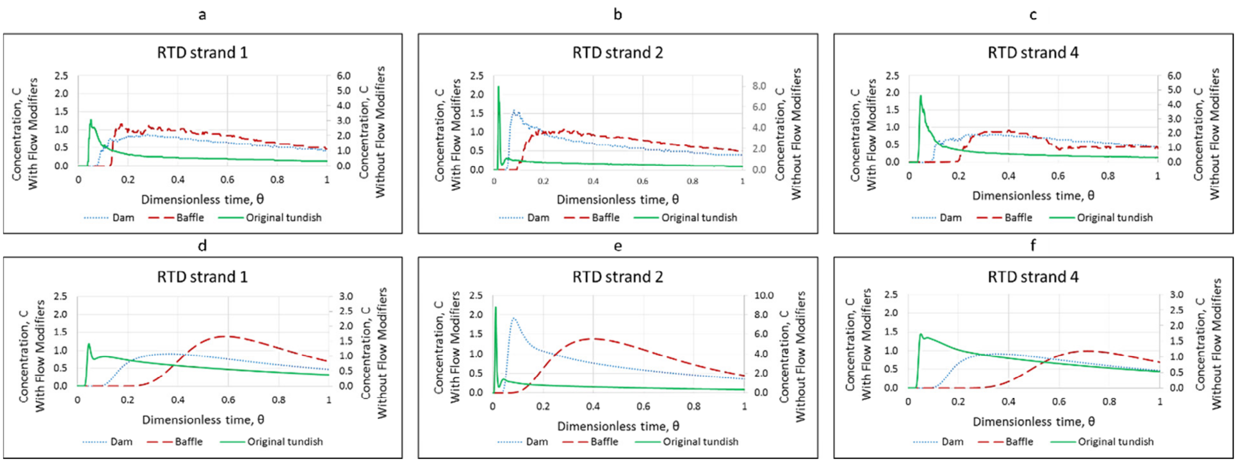


Fig. 8. RTD curves from the CFD simulations from (a) outlet-1, (b) outlet-2, (c) outlet-3; The experiment results from the physical water model at (d) outlet-1, (e) outlet-2, (f) outlet-3.

From Fig. 8 (b) and (e), it can be seen from the green solid lines that the RTD from outlet-2 in the bare tundish (original tundish) is critical, where the peak of RTD curve is quite close to the start point at the left side of the graph which mean the residence flow time is very short. Both values of the minimum residence time (θ_{start}) and the maximum residence time (θ_{peak}) of the RTD curve is very low, while the dimensionless concentration (C) at θ_{peak} is very high. This flow characteristic provides high opportunity of the short-circuit flow and the contamination of inclusion in the casting mold. The RTD from outlet-2 after fitting with dam and baffle can be seen from the blue and red curves in Fig. 8 (b) and (e). It can be seen that the θ_{start} and θ_{peak} of both curves are increased and move to the right side of the graph, which means that they can prevent the short-circuit flow and promote inclusion removal. The tundish with baffle shows better results than that with the dam.

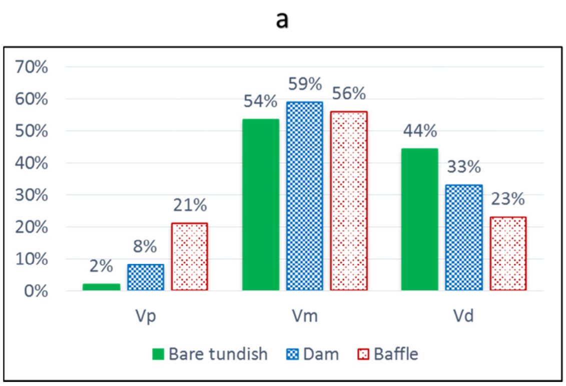
Fig. 8 (a) and (d) show the RTD curves from outlet-1. Fig. 8 (c) and (f) show the RTD curves from outlet-4. All the results have similar behavior as of those from outlet-2 but less critical, where θ_{start} and θ_{peak} of the RTD curves of outlet-1 and outlet-4 of the bare tundish is not as low as of outlet-2. The fitting with dam and baffle also prolong the RTD curves which improve the flow characteristic.

The mean residence time (θ_{mean}) is the average of residence flow time in tundish from inlet to outlet. Both results from CFD simulation and water model experiments show that with the installation of dam and baffle the θ_{mean} can be increased for more than 20%. The key indicators for the flow characterization such as θ_{start}, θ_{peak} and θ_{mean} of the RTD curves are summarized in Table 2.

Table 2. Dimensionless time points on the RTD curves from CFD simulation and experiment water model results.

Tundish model / outlet no.	θ_{Start}		θ_{Peak}		θ_{Mean}	
	CFD	Experiment	CFD	Experiment	CFD	Experiment
Bare tundish / outlet-1	0.03	0.04	0.05	0.05		
Bare tundish / outlet-2	0.01	0.02	0.01	0.02	0.62	0.65
Bare tundish / outlet-4	0.03	0.04	0.05	0.05		
Tundish with dam / outlet-1	0.10	0.07	0.35	0.21		
Tundish with dam / outlet-2	0.04	0.05	0.09	0.08	0.75	0.77
Tundish with dam / outlet-4	0.10	0.09	0.37	0.22		
Tundish with baffle / outlet-1	0.24	0.13	0.59	0.19		
Tundish with baffle / outlet-2	0.10	0.09	0.39	0.21	0.84	0.79
Tundish with baffle / outlet-4	0.30	0.20	0.72	0.41		

The flow characteristics can also be classified by separating the flow volumes into two types: the active flow volume and the stagnant flow or dead volume (V_d). Active flow volume consists of the plug flow volume (V_p) and the well-mixed volume (V_m). The percentage of volume fractions of all types were calculated based on the RTD curves from CFD simulation and the physical water model as shown in Fig. 9 (a) and (b).



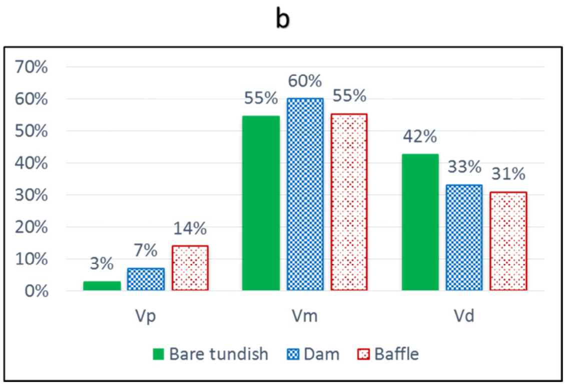


Fig. 9. Volume fraction of the flow in all tundish models obtained from (a) CFD simulation, (b) the physical water model.

From Fig. 9 (a) and (b), the volume fraction of stagnant flow (Vd) is decreased after fitting the flow modifiers. The flow modifiers also promote the volume fraction of active flow. These mean the flow characteristics were improved, since the small dead volume indicates the better of temperature homogeneity.

5. Conclusion

The physical water model and the CFD simulation can be used for the prediction of flow behavior in tundish. The results from both cases are in agreement with minor difference. For the tundish at Millcon steel PLC, the flow at middle outlet-2 is critical and has the lowest of the minimum residence time (θ_{start}). The flow modifiers improve the residence time, especially the θ_{start} from the middle outlet-2 which are increased for more than 2 times. This will promote the inclusion removal potential for the molten steel.

The future work of this research would be the investigation of the effect of longer residence time on the temperature and flow characteristics of the molten steel under non-isothermal simulation conditions.

Acknowledgements

This research was funded by King Mongkut's University of Technology North Bangkok (Contract no. KMUTNB-60-GOV-17 and Contract no. KMUTNB-60-GEN-20) and Millcon Steel PLC (Contract no. TGGS-RAS-002/2559).

References

[1] A. Kumar, D. Mazumdar, S.C. Koria, Modeling of fluid flow and residence time distribution in a four-strand tundish for enhancing inclusion removal, ISIJ Int. 48 (1) (2008) 38–47.
[2] Y. Sahai and T. Emi, Tundish Technology for Clean Steel Production, World Scientific, 2008, 316 p.
[3] D. Mazumdar and R. I. L. Guthrie, The Physical and Mathematical Modelling of Continuous Casting Tundish Systems, ISIJ Int., Vol 39 (1999), No. 6, pp. 524-547
[4] K. Chattopadhyay, M. Isac, R. I. L. Guthrie, Physical and Mathematical Modeling of Steelmaking Tundish Operations: A Review of the Last Decade, ISIJ International, 50 (2010) 331–348.
[5] A. Kumar, S. C. Koria, D. Mazumdar, An Assessment of Fluid Flow Modelling and Residence Time Distribution Phenomena in Steelmaking Tundish Systems, ISIJ International, Vol. 44 (2004), No. 8, pp. 1334–1341
[6] 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
[7] P. Kowitwarangkul, A. Harnsihacacha, Tracer Injection Simulations and RTD Analysis for the Flow in 3-Strands Steelmaking Tundish, Key Engineering Materials, Vol. 728, pp. 72-77, 2017
[8] K. Raghavendra, S. Sarkar, S.K. Ajmani and M.B. Denys a, M.K. Singh, 2013, Mathematical modelling of single and multi-strand tundish for inclusion analysis, Applied Mathematical Modelling, 37 (2013) 6284–6300
[9] J. H. Cloete, G. Akdogan, S. M. Bradshaw, D. K. Chibwe, Physical and Numerical Modelling of a Four-Strand Steelmaking Tundish Using Flow Analysis of Different Configurations, P Pyro Mod Con, (2014) 157-169.
[10] 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 & Fluids, Vol. 24, 1995, Issue 3, p. 227-238