Available online at www.sciencedirect.com

ScienceDirect

Materials Today: Proceedings 5 (2018) 9220–9228

materialstoday: PROCEEDINGS

www.materialstoday.com/proceedings

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

Physical water model and CFD studies of fluid flow in a single strand tundish

Anawat Harnsihacacha, Adisorn Piyapaneekoon, Pruet Kowitwarangkul*

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

Abstract

Flow characteristics in a continuous casting tundish is one of the key parameters producing a high quality clean steel. In this study, the fluid flow analyses of a single-strand tundish model were carried out using the commercial CFD software ANSYS FLUENT 14.0 and experimentally verified by physical water model. The realizable k-ε equation was utilized to model turbulent phenomenon. The behavior and performance of the flow in tundish with and without flow modifiers were investigated by the residence time distribution (RTD) curves which were obtained from the tracer concentration measurement at the outlet. The results from CFD are in agreement with the experiment results. The results show that the flow modifier plays an important role in increasing residence time and enhancing flow performance which could feasibly promote the inclusion removal potentials in the tundish process. The peak and the minimum residence time of RTD curves of the tundish model with flow modifiers were improved for more than 20%. With the comparison of the four flow modifier configurations (bare tundish, dam, baffle and turbostop) in the current tundish model, it could be concluded that the turbostop could provide the optimal flow characteristic in which could improve the level of inclusion removable.

Keywords: Single strand tundish; Residence time distribution; Computational fluid dynamics; Physical water model

1. Introduction

In the past few decades, the demand for high-grade steels has been increasing consistently. Steel cleanliness, which refers to the minimum amount of non-metallic inclusions, is a key factor of steel quality. In a continuous

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

casting process, a tundish traditionally refers to an intermediate vessel placed between the ladle and the casting mold, designed to reserve and distribute molten steel to each mold. With continuing demand on superior steel quality, tundish has evolved into a purify reactor for liquid steel refining. The tundish is a refractory-lined channel consisting of an inlet and outlet sections and sometimes is furnished with flow modifiers, e.g. dams, baffles and turbostop. These flow modifiers increase the residence time of the flow in tundish and improve the steel cleanliness by letting the non-metallic inclusion float and be captured by the slag at the top surface of the molten steel [1,2].

Considerable efforts to enhance the inclusion removal performance of continuous casting tundish systems have been carried out through the numerous physical and mathematical modelling studies embodying both industrial and water model tundishes [2]. The review article on tundish numerical modelling [3] showed that most of researchers had used the k-ε model to model turbulent phenomenon of the flow in the CFD simulation of tundish. Some researchers [4] found that applying the Reynolds stress model is somewhat superior to k-ε model, but it uses more CPU time. The results from the research [5] showed the example of the tundish modification which prolong the residence time and promote the inclusion removal by floatation. With the implementation of the flow modifiers in tundish, the streamlines of molten steel had more opportunity to contact with the steel-slag interface at the top layer which also enhanced the inclusion removal [6]. Tundish without flow control may have a significant short-circuit flow [7], which results in the increasing of inclusion contamination in the casting product. The water model can be used for the physical simulation, because the kinematic viscosity of water and liquid steel at 1600 ^{o}C are nearly equivalent. Applying Froude similarity criterion, the results from reduced-scale water model can be represented for the real process [1]. In the study [7], 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.

In the earlier researches [8,9] the flows in tundish were investigated using numerical simulation. The results show that the flow modifiers improve flow characteristics and the residence time was increased. However, those numerical simulation results were compared only with some experiment results from the literatures. In the current study, the new tundish water model station is setup in the laboratory and the CFD simulation results were compared and validated with the experiment results from the physical water model. The aim of the present research is to study the flow behavior inside a single strand tundish model and to improve the inclusion removal potential and flow behavior by the implementation of flow modifiers.

2. Research methodology

2.1. Numerical simulation setup

In the current study, the flow in tundish model is numerically simulated using the commercial software ANSYS FLUENT 14.0. Fig.1 (a) shows the geometry and boundary conditions of the 1-strand tundish models. Constant velocity, symmetry and constant pressure conditions are applied as the boundary conditions at inlet, top surface and outlet respectively. An unstructured computational mesh of around 0.7 - 1 million cells is used for the simulation. The mesh is shown in Fig.1 (b).

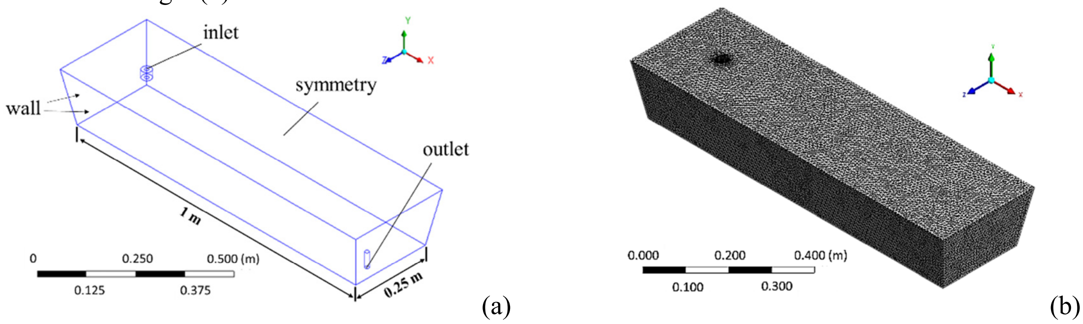


Fig.1. (a) Boundary conditions and (b) mesh elements inside the single strand tundish model.

Water level of 0.173 are used for the simulation and experimental setting up. Inlet and outlet diameter are 0.012 m. (outlet area is 1.1304×10^{-4} m^2). Velocity at outlet is 1.753 m/s and flow rate of water is 11.891 liter/minute. The numerical simulations are carried out on the basis of the Reynolds-Averaged Navier-Stokes (RANS) modelling. The realizable k-ε model from Shih et al. [10] 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) isothermal condition, (2) 3-D steady state and (3) transient during tracer injection. The basic continuity equations describing the fluid flow phenomena in the current study are as follows:

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

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

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

where ρ is the density (kg m^{-3}), u is the flow velocity (m s^{-1}), t is the time (s), 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 and equation (3) describes the effective stress tensor.

2.2. Tracer injection simulation and RTD analysis

The flow characteristics of tundish without and with flow modifiers are analyzed by the residence time distribution (RTD) curve. In numerical simulation, a tracer which has same properties as of domain fluid is 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. Steady state flow of the domain fluid is firstly calculated, afterward, a tracer injection is simulated using species transport model with transient mode. After stop injection, the mixed flow of tracer and the domain fluid is simulated with transient mode for 20 minutes. For the flow characterization, the first step is to derive the dimensionless C-curve for the tundish. Y. Sahai et al. [1] described the calculation method as following:

The dimensionless time, θ, was calculated as Eq. 4:

$$\theta = \frac{t}{\bar{t}} \tag{4}$$

where $\bar{t}$ is the theoretical mean residence time as Eq. 5:

$$\bar{t} = \frac{V}{Q} \tag{5}$$

The dimensionless concentration of strand *i* (the outflow at outlet *i*) can be calculated as Eq. 6:

$$C_i = \frac{C_i V}{M} \tag{6}$$

where V is the total volume of the tundish, C_i is the concentration at the outlet *i*, and M is the total amount of tracer injected.

The mean residence time of the flow, t_{mean}, is calculated as Eq. 7:

$$t_{mean} = \frac{\int_0^\infty tC(t)dt}{\int_0^\infty C(t)dt} \tag{7}$$

In the second step, the tundish performance can be classified by separating the flow volumes into three types: the plug flow (V_p), the well-mixed volume (V_m), and the dead volume (V_d) as Eq.8-10:

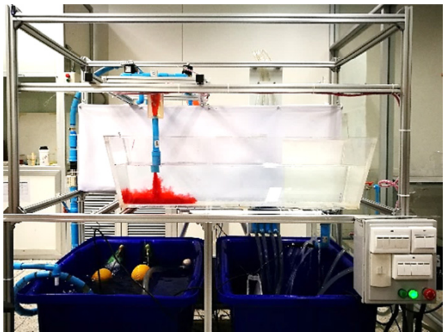
$$V_p = \theta_{min} \tag{8}$$

$$V_d = 1 - \frac{Q_a}{Q} \times \theta_{mean} \tag{9}$$

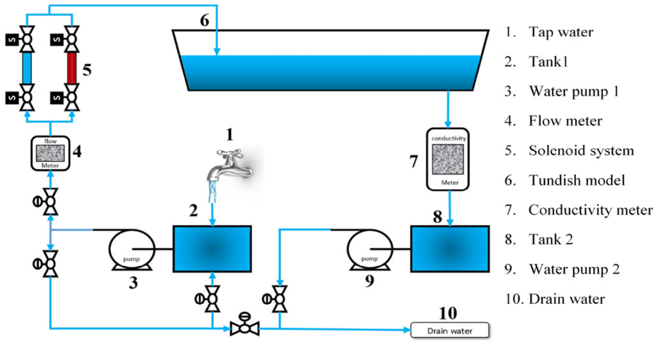
$$V_m = 1 - V_p - V_d \tag{10}$$

2.3. Experimental setup

Fig. 2 (a) shows the physical water model of single strand tundish. Fig. 2 (b) shows the schematic diagram of water modeling system setup used to analyze the tracer injection and the residence time distribution. The height of water lever was controlled to be constant at 17.3 cm. The system consists of a solenoid valve tracer injector combined with water feeding system. Normally, in many experiments of tundish water models, the tracer was injected using syringe and this can cause the change of flow velocity of the domain fluid during injection time. The solenoid valve tracer injector designed in the current study can avoid this flow velocity change. 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 and injected to the flow system by open the solenoid valve. The tracer concentration was measured and recorded every 1 second at the outlet by using electric conductivity meter.



(a)



(b)

Fig. 2. (a) the physical water model of single strand tundish (b) the schematic diagram of water modeling system setup used to analyze the tracer injection and the residence time distribution.

2.4. Flow modification

In both cases of the experiment tundish model and the simulation model, there are 3 designs of the flow modifiers as shown in Fig. 3. For the tundish with dam, the dam height is 10 cm from bottom and the thickness is 1.5 cm. For tundish with baffle, there are 8 holes on the baffle plate which has 1.5 cm thickness. For tundish with turbostop, the inner dimension of the box is 13x13x3 cm^3.

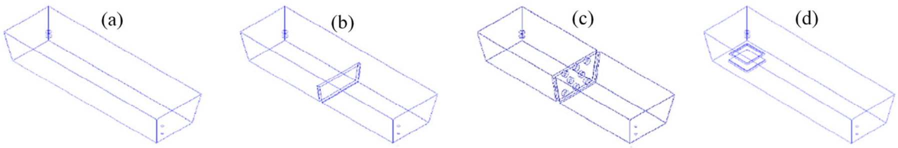


Fig. 3. Flow modification: (a) bare tundish, (b) dam, (c) baffle and (d) turbostop.

2.5. Mesh dependence

Mesh dependence effect on the simulation results is investigated in the current study. In case of tundish with turbostop, it was found during study that the mesh with 0.9 million elements might not be enough to provide a precise simulation result when compared to the experiment result. In this case, the finer mesh elements are required. Four cases of simulation with different mesh are conducted. In Fig.4, the pictures below show the mesh at the cross section plane in turbostop zone of tundish model. Fig. 4. (a), (b) shows the mesh with 0.9 and 4 million cell elements in total volume of tundish model. Fig. 4 (c) shows the mesh with 8 million cell elements in half of total volume of tundish model and the simulation is conducted in longitudinal symmetry half volume of tundish model. Fig. 4 (d) shows the mesh with 1.5 million cell elements in half of total volume of tundish model where the volume is separate into 2 parts and the 2nd part in the inlet zone has finer mesh element. The numerical simulations are conducted on these 4 cases of mesh with same calculation condition.

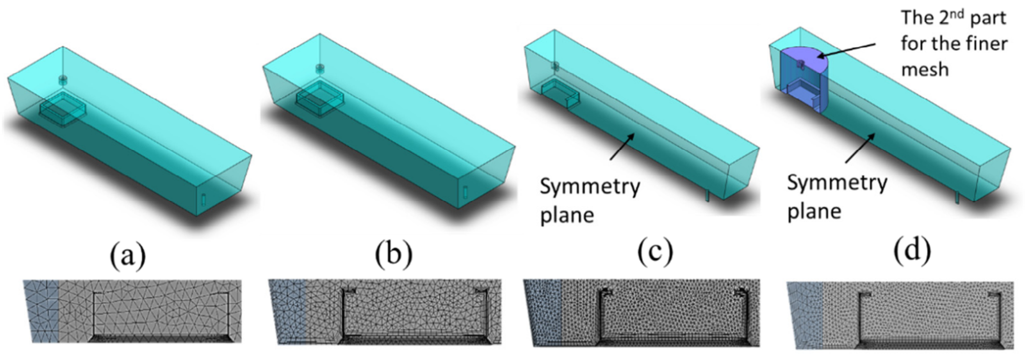


Fig. 4. Mesh with (a) 0.9 million elements, (b) 4 million elements, (c) 8 million elements – symmetry (d) 1.5 million elements – symmetry – 2 parts.

3. Results and discussion

3.1. Velocity flow field

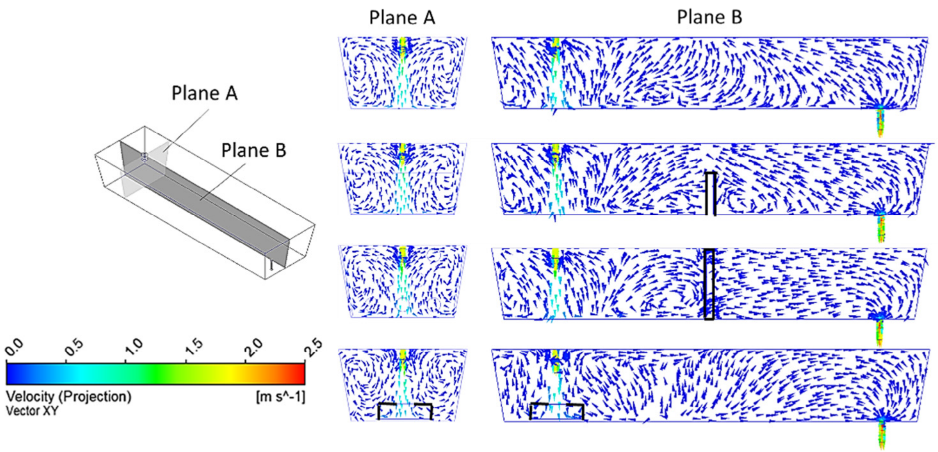


Fig. 5. Velocity flow field of (a) bare tundish, (b) tundish with dam, (c) tundish with baffle and (d) tundish with turbostop.

Velocity flow field in the bare tundish (tundish without flow modifier) and the tundish with flow modifiers such as dam, baffle and turbostop are illustrated in the 2-D cross section plane A and plane B as shown in Fig. 5. The arrow heads and color represent the direction and velocity of the flow. The flow structure of the two counter-rotating toroidal vortices can be captured on the plane A. In cases (a)-(c), the fluid flows from inlet to bottom of tundish with high velocity, separate to both side walls and flow to the top surface. In case (d), the vortices direction of the tundish model with turbostop are opposite to the other 3 cases. Plane B shows the change of the velocity flow field which covers the whole fluid from inlet to outlet.

3.2. Streamline

The streamlines in the tundish models are simulated by the random injection of sampling particles which have the same properties as of the fluid. Streamlines of the tundish before and after fitting with flow modifiers are shown in Fig. 6. The color of streamline represent individual pathline of fluid flow from inlet to outlet. Flow structure of fluid in bare tundish with counter-rotating toroidal vortices can be ssen and this flow pattern will gradually disappear when it flow to outlet zone. After fitting the tundish with dam or baffle, flow pattern was changed. In tundish with dam, the vortices occur at both side of dam. In tundish with baffle, the vortices occur at inlet side. After liquid flow pass baffle, liquid flow structure become less vortex than before pass through the baffle. In tundish with turbostop, the vortices occur only near the turbostop.

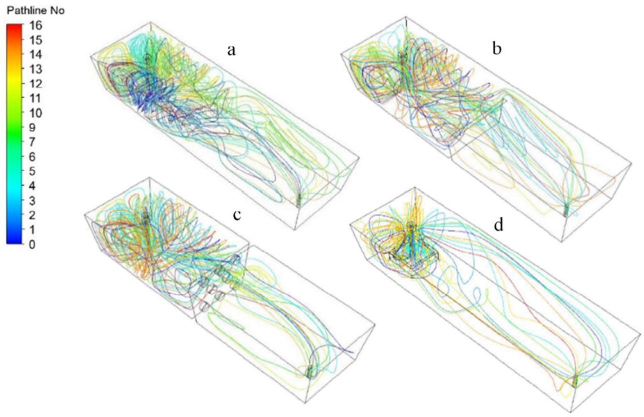


Fig. 6. Streamlines of fluid flow inside the tundish model: (a) bare tundish, (b) tundish with dam (c) tundish with baffle and (d) tundish with turbostop.

3.3. Turbulence kinetic energy

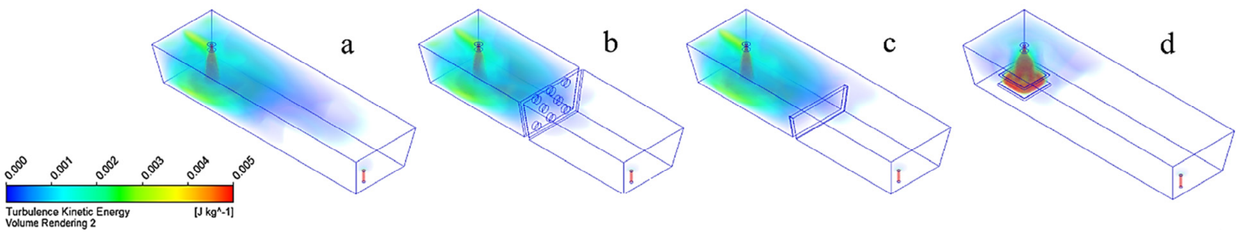


Fig. 7. Turbulance kinetic energy (a) bare tundish, (b) tundish with dam, (c) tundish with baffle and (d) tundish with turbostop.

The turbulence kinetic energy in the whole volume of each tundish model is shown in Fig. 7. It can be seen that the flow modifier, especially turbostop, changes the nature of the turbulence. High turbulence kinetic energy occur in the volume near the inlet, where more turbulence flow takes place. In case of the bare tundish, the tundish with dam and baffle, a high turbulence zone is formed near the flow inlet and the turbulence zone expand to around half of tundish volume. In case of the tundish with turbostop, the high-turbulence zone is intense in the zone inside turbostop and the turbulence zone from the whole tundish volume is smaller than the other cases.

3.4. Tracer injection

Tracer injection simulation is one way to reveal the flow characteristic of fluid flow inside tundish. The comparisons between simulation results and visual observation of red dye tracer mixing phenomena in the water model for the bare tundish, the tundish with dam, the tundish with 8-hole baffle and the tundish with turbostop are shown in Fig. 8 and 9. Tracer mass fractions in the fluid at 1 and 3 seconds after injection are shown with red-black color scale. The simulation results are in agreement with the water model results. In the tundish, the incoming tracer from the inlet that quickly enters the outlet represents a short circuiting flow. Flow of tracer in the bare tundish move freely and fast toward outlet, some amount of tracer might flow to outlet as a short circuit. The use of flow modifiers, especially turbostop (Fig. 8d and 9d), changes the flow patterns significantly and is able to decrease the short circuit phenomena.

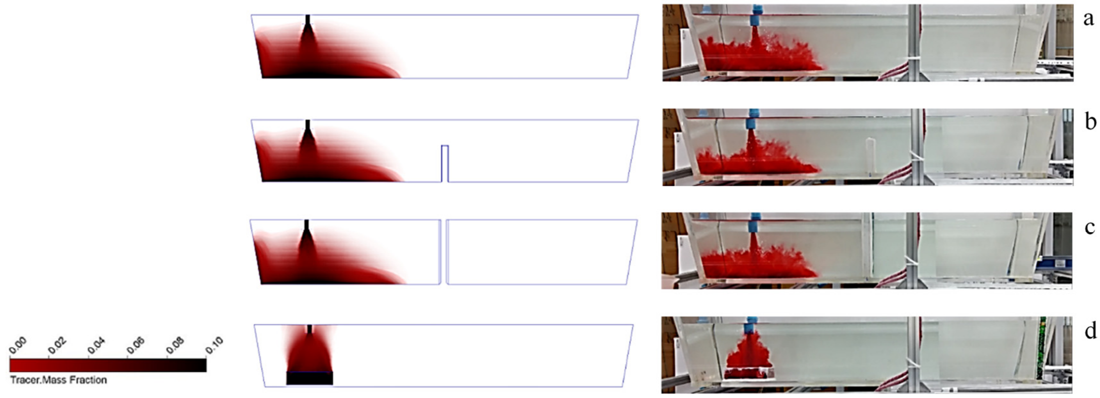


Fig. 8. Tracer flow after 1 second of injection in (a) bare tundish, (b) tundish with dam, (c) tundish with baffle and (d) tundish with turbostop.

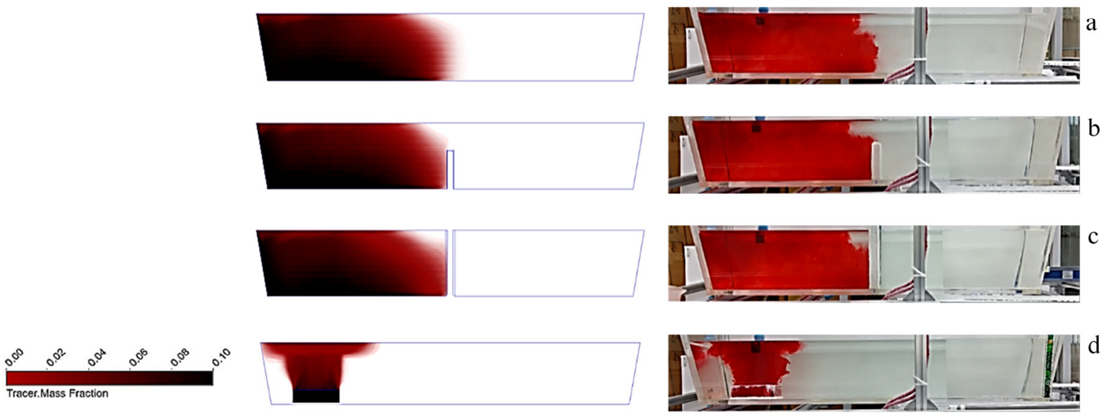


Fig. 9. Tracer flow after 3 seconds of injection in (a) bare, (b) tundish with dam, (c) tundish with baffle and (d) tundish with turbostop.

3.5. RTD curves

The residence time curves which show the dimensionless mass fraction of the tracer against dimensionless time (θ) for each tundish models from the CFD simulation and water model experiment results are represented in Fig.10. The key indicators for the flow characterization such as θ_{start}, t_{start}, θ_{max}, t_{max}, θ_{mean}, t_{mean} of the RTD curves, plug volume (V_p), well-mixed volume (V_m) and dead volume (V_d) flows from the CFD simulation and water model experiments are summarized in Table 1 and 2. The results between the CFD simulation and the experiment results shows good agreement with minor difference.

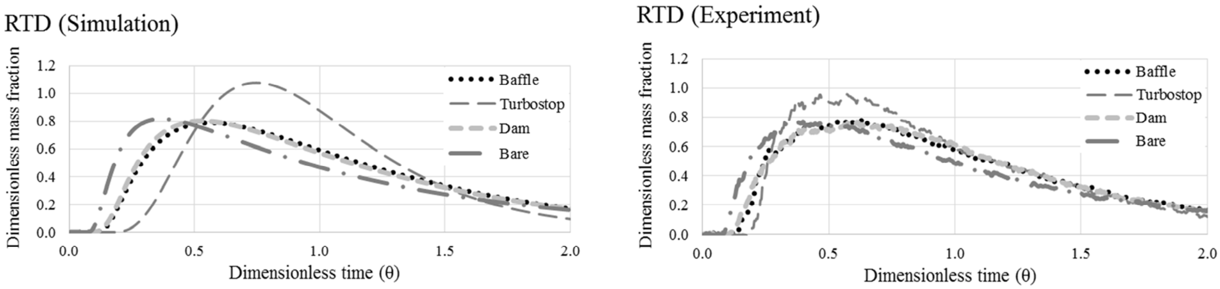


Fig. 10. Residence time distribution (RTD) curves for each tundish model: (left) simulation results, (right) experiment results.

From the results shown in Table 1 and 2, all of the flow time indicators are improved after the implementation with flow modifiers. The tundish with turbostop provide the optimal flow characteristic, where θ_{start} are improved for more than 2 times. The mean residence time (t_{mean} and θ_{mean}) of all tundish with flow modifiers are improved for more than 10%. These are indicators that the prevention of short-circuit flows and the floatation of inclusions are promoted. Moreover, with the current design of the flow modifiers, the percentage fractions of stagnant flow or dead volumes (V_d) are improved and decreased.

Table 1. Parameters of RTD curves and fraction flow from the CFD simulation results.

Tundish model	Residence time						Dispersed plug flow	Ideal mixing flow	Dead flow
	θ_{start}	t_{start} (sec)	θ_{max}	t_{max} (sec)	θ_{mean}	t_{mean} (sec)	(V_p)	(V_m)	(V_d)
Bare	0.08	20	0.36	92	0.83	211	%8.10	%63.60	%28.30
Dam	0.12	30	0.53	133	0.91	227	%12.00	%68.20	%19.80
Baffle	0.12	30	0.42	104	0.92	230	%12.00	%69.50	%18.40
Turbostop	0.20	50	0.75	190	0.95	240	%19.95	%71.79	%8.36

Table 2. Parameters of RTD curves and fraction flow from the water model experiment results.

Tundish model	Residence time						Dispersed plug flow	Ideal mixing flow	Dead flow
	θ_{start}	t_{start} (sec)	θ_{max}	t_{max} (sec)	θ_{mean}	t_{mean} (sec)	(V_p)	(V_m)	(V_d)
Bare	0.08	20	0.48	121	0.85	214	7.92%	64.97%	27.12%
Dam	0.12	30	0.61	154	0.91	227	11.97%	69.03%	19.00%
Baffle	0.14	34	0.59	146	0.91	228	13.64%	67.74%	18.62%
Turbostop	0.18	45	0.58	146	0.88	222	17.87%	65.16%	16.98%

3.6. Mesh dependence result

The 4 cases of simulation with different mesh element number are conducted in the tundish model with turbostop. Fig. 11 shows the RTD curve of the 4 cases of simulation: 0.9 million cells, 4 million cells, 8 million cells – symmetry and 1.5 million cells – symmetry – 2 parts. In case of 8 million cells, the CFD simulation with θ_{start} of 0.2 is the most correspond to the RTD curve of the experiment water model where θ_{start} is 0.18. However, with 8 million cells mesh, the computer calculation time is quite so long for around 6 weeks. In case of 1.5 million cells – symmetry – 2 parts, the CFD simulation has the sufficient precise result (θ_{start} = 0.24) with the efficient computer calculation time for around 1 week.

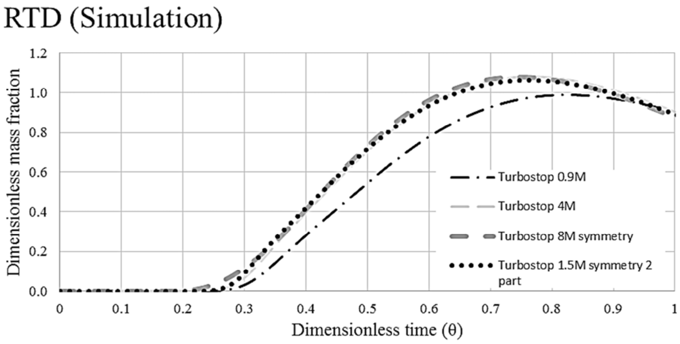


Fig. 11. Residence time distribution (RTD) curves of tundish model with turbostop and different mesh resolution: 0.9 million elements, 4 million elements and 8 million elements – symmetry, 1.5 million elements – symmetry – 2 parts.

4. Conclusion

The physical water model and the CFD simulation can be used for the prediction of flow behavior in tundish. Flow modifiers can improve flow characteristic in tundish models. The short-circuit flows are reduced and the residence time distribution are improved, which could feasibly promote the inclusion removal potentials in the tundish process. With 0.9 million cells of mesh elements, all of the CFD simulation results are in good agreement with the physical water model results (for example, the difference of t_{mean} between both results are less than 2%), except in case of the tundish model with turbostop, where the finer mesh applied with two separated mesh zone in the symmetry half volume is required to obtain the precise simulation results and the highest efficiency of CPU calculation time.

For the tracer injection test results from the water model experiments and CFD simulations, the peak (θ_{max}, t_{max}) and the minimum residence time (θ_{start}, t_{start}) of RTD curves of the tundish model with flow modifiers are improved for more than 20%. With the comparison of the four flow modifier configurations (bare tundish, dam, baffle and turbostop) in the current tundish model, turbostop provides the optimal flow characteristic in which could improve the level of inclusion removable.

Acknowledgement

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.

References

[1] Y. Sahai and T. Emi, Tundish Technology for Clean Steel Production, World Scientific, 2008, 316 p.
[2] 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
[3] 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.
[4] 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
[5] B. Buĺko and J. Kijac, "Optimization of Tundish Equipment", Acta Metallurgica Slovaca, Vol. 16, 2010, No. 2, p. 76-83
[6] M. Warzecha, Computational Fluid Dynamics Technologies and Applications, InTech, 2011.
[7] 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.
[8] 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
[9] 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
[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