MODELING AND NUMERICAL SIMULATION OF FLUID FLOW OF A STEEL CONTINUOUS CASTING TUNDISH

J. R. S. Rocha, F. Marcondes
Department of Metallurgical and Materials Engineering
Pici Campus, Block 729, 60440-554, Fortaleza-Ce.
jrsousaufc@gmail.com

ABSTRACT

Currently, the continuous casting process is the most used technique to produce steel. Being an inherently component of the caster machine, the tundish has been designed to be not only an intermediate vessel between the ladle and the mold, but also a device to remove inclusions. The physical model for fluid flow into the tundish is very complex, therefore, analytical solutions are not available. Hence, Computational Fluid Dynamics emerges as an attractive alternative. The main goal of the present study is to analyze the fluid flow into an actual tundish. Based on the performed simulations, some modifications in the geometry of the tundish were proposed in order to improve the steel's quality. For solving the governing equations, the ANSYS CFX software was used. Simulations were performed using water as working fluid for a turbulent flow in a 3D tundish. The results were presented in terms of residence time distribution curves.

Keywords: Numerical Simulation, Continuous Casting Tundish, RTD, ANSYS CFX, EbFVM.

INTRODUCTION

Currently, the continuous casting process is the most used technique to produce steel. In order to attain steel cleanliness, one of the most important component devices employed in this process is the so-called tundish. In the last decades, the tundish has been designed to be not only an intermediate vessel between the ladle and the mold, but also an inclusion removal and a metallurgical reactor. With the aim to increase the quality of the steel, some flow control devices have been added to the tundish to control the flow pattern. From the fluid flow modifiers we can cite dams, weirs, stopper rods, turbulence inhibitors, and gas injectors.

The steel fluid flow into the tundish is promoted by the inlet nozzle. At the inlet region of the tundish the flow regime is mostly turbulent, while far from the inlet (bulk region), turbulence decreases progressively, and hence inclusions can be removed(1). It is observed that two-equation turbulence models are the first ones preferred by industry for modeling turbulence flows. Also, two-equation eddy viscosity models are still the first choice for CFD calculations, with the standard $k-\varepsilon$ model(2) and $k-\omega$ model(3) being the most widely used.

Considering the importance of liquid steel to metallurgy industry, several researchers have carried out investigations in both experimental and numerical area, concerning the main aspects that govern the fluid flow into the tundish(1,4). One way to assess information about the flow field in a tundish, either numerically or experimentally is by injecting a tracer in the incoming flow and monitoring its concentration at exit. By plotting this concentration against time, one obtains the Residence Time Distribution (RTD) curve. The combined or mixed model has been quite used to the assessment of RTD curves, and consequently it has been responsible for calculating the plug flow, mixed flow, and dead volumes into the tundish(5).

ANSYS CFX uses the Element-based Finite-Volume method (EbFVM) to discretize the partial differential equations. Moreover, Finite-Volume Method is the most used method in commercial packages, since it is the only approach that can guarantee local conservation of the physical quantities (mass, momentum, energy)(6).

Therefore, based on numerical simulation, the main objective of this study is to evaluate the flow field within a continuous casting tundish of the Gerdau Company. In addition, we propose some modifications by making use of dams and weirs.

MATHEMATICAL MODEL

Equations

The fluid flow into the tundish can be modelled by the mass and momentum equations. For modeling turbulence quantities, the two-equation $k-\varepsilon$ model was used. After averaging the continuity and momentum equations, and using the Boussinesq eddy-viscosity approximation to relate the Reynolds stresses to the strain rate of the mean motion, one obtains:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho U_j\right)=0, \tag{A}$$

$$\frac{\partial}{\partial t}\left(\rho U_i\right)+\frac{\partial}{\partial x_j}\left(\rho U_i U_j\right)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\right], \tag{B}$$

where ρ is fluid density, U_j are velocity components, p is pressure, and μ_{eff} is the effective viscosity accounting for turbulence, which is defined as:

$$\mu_{eff}=\mu+\mu_t, \tag{C}$$

where μ is the dynamic viscosity and μ_t is the turbulent viscosity.

$k-\varepsilon$ model assumes that the turbulence viscosity is related to the turbulence kinetic energy k and the turbulence eddy dissipation ε by the following relation:

$$\mu_t=C_\mu \rho \frac{k^2}{\varepsilon}, \tag{D}$$

The values of k and ε are calculated via the two transport equations:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}\left(\rho U_j k-\frac{\mu_{eff}}{\sigma_k}\frac{\partial k}{\partial x_j}\right)=P_k-\rho\varepsilon, \tag{E}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_j}\left(\rho U_j \varepsilon-\frac{\mu_{eff}}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right)=\frac{\varepsilon}{k}\left(C_{\varepsilon 1}P_k-C_{\varepsilon 2}\rho\varepsilon\right), \tag{F}$$

where C_μ, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k, and σ_ε are constants taken from Launder and Spalding(2). P_k is the turbulence production due to viscous forces and is given as:

$$P_k=\mu_t\frac{\partial U_i}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right). \tag{G}$$

To obtain the RTD curve, it is necessary to solve the tracer diffusion equation that is described by:

$$\frac{\partial}{\partial t}(\rho C)+\frac{\partial}{\partial x_j}\left(\rho U_j C\right)=\frac{\partial}{\partial x_j}\left(D_{eff}\frac{\partial C}{\partial x_j}\right), \tag{H}$$

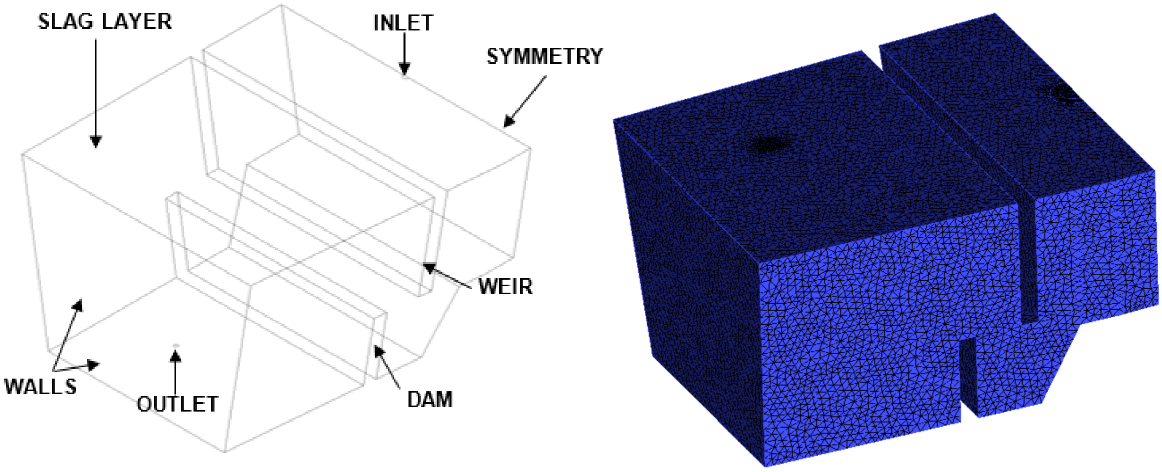
where *C* is the tracer concentration and D_{eff} is the effective kinematic diffusivity, which is defined as:

$$D_{eff}=\rho D+\frac{\mu_t}{S_c}, \tag{I}$$

where *D* is the kinematic diffusivity and S_c is the turbulent Schmidt number.

Geometry and mesh configuration

The present study employed the commercial software ANSYS CFX to perform numerical analysis. In the pre-processing step, the geometry as well as the mesh were created using the ANSYS ICEM CFD software. Three cases were studied, namely, case I (actual bare tundish), case II (tundish with dam), and case III (tundish with dam and weir). Fig. 1 shows the computational domain for case III. In addition, an unstructured mesh for the same case, which is composed of tetrahedral and prism elements, is also depicted in the same figure.



(a) (b)

Figure 1: Half tundish geometry and mesh. (a) Tundish with dam and weir, (b) Unstructured mesh for case III.

Table 1 shows the operating physical parameters for the actual tundish configuration (case I) in addition to the proposed changes in tundish geometry by adding dam (case II) and dam and weir (case III).

Table 1: Physical parameters for numerical analysis.

Case I - Volume	0.319 m³
Case II - Volume	0.313 m³
Case III - Volume	0.303 m³
Inlet Diameter	0.0215 m
Outlet Diameter	0.0125 m
Weir Height	0.323 m
Dam Height	0.215 m
Inlet Velocity	3.57 m/s

Boundary and initial conditions

At the inlet, we prescribed the normal velocity and turbulence intensity as 3,57 m/s and 5%, respectively. At the outlet, we prescribed a zero Pa for pressure, and zero normal gradient for all other variables used. No-slip wall condition as well as wall functions were also applied. For modeling the free surface of steel bath, free-slip wall condition was set. In order to take advantage of geometry symmetry, only half of the tundish configuration was used and hence, at symmetry plane values of the normal velocity component as well as scalar variable gradients normal to the boundary were set to zero. The flow field obtained from steady-state analysis was used for solving the tracer equation in order to obtain the RTD curves. As initial condition, we set the tracer concentration equal to zero. Zero gradient boundary conditions for all tundish regions were applied, except for the inlet region, where a pulse tracer injection was prescribed.

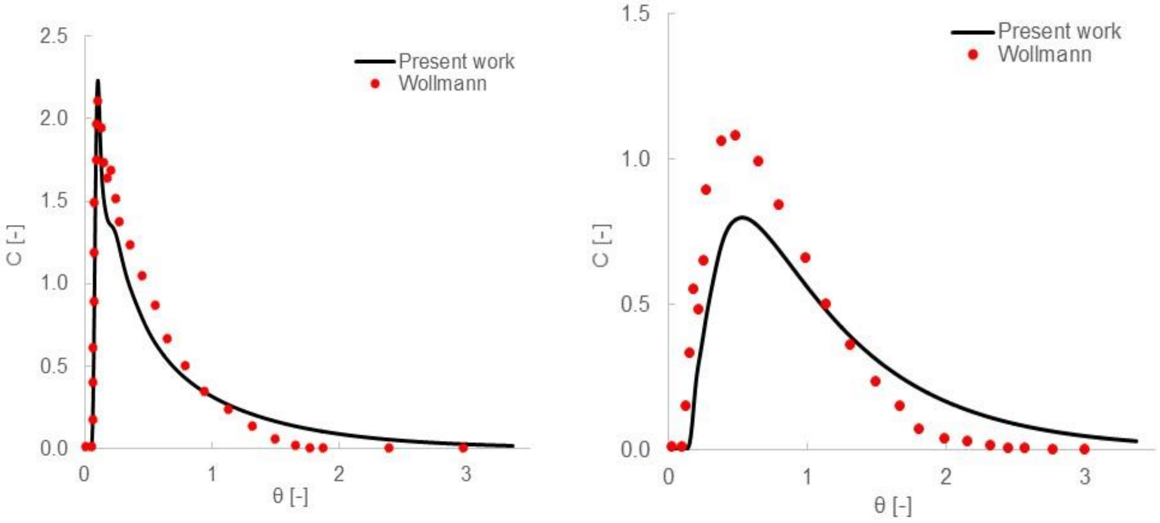
Solution procedure

In ANSYS CFX, the mass and momentum equations are solved coupled for pressure and velocity components using the Element-based Finite-Volume Method. After solving the variables, the $k-\varepsilon$ model is solved using a segregated approach. After the velocity and pressure fields reach the steady-state regime, the concentration equation is solved to obtain the tracer path into the tundish. The algebraic system of linear equations obtained after discretization are solved by using an algebraic multigrid methodology called the additive correction multigrid method(7). Convergence was achieved when the RMS (root mean square) residuals were equal or smaller than 10^{-5} for steady state analysis and 10^{-6} for transient analysis.

RESULTS AND DISCUSSION

A mesh refinement study was performed for all case studies investigated. The final simulation meshes for validation case, case I, case II, and case III were composed of 259k, 132k, 137k, and 144k nodes, respectively. A fine mesh was mainly concentrated where higher velocity gradients were expected to appear (boundary layer) as well as at regions of high turbulence, such as inlet and outlet zones.

In order to obtain numerical validation on the methodology employed by the present work, a validation case was also performed. The results presented in Fig. 2 were taken from Wollmann(8). From Fig. 2, it is observed that the C-curve for two exit nozzles obtained from numerical simulation, showed a good agreement with the physical analysis.



(a) (b)

Figure 2: Residence time distribution for numerical and experimental(8) works. (a) Central exit nozzle, (b) lateral exit nozzle.

Table 2 presents comparisons for minimum and mean residence time. Minimum residence time showed a good agreement with the experimental work, whereas the mean residence time despite being less accurate, showed an acceptable agreement as well.

Table 2: Minimum and mean residence time for numerical and experimental(8) works.

	θ_{min} [-]	θ_{mean} [-]
Present work (Numerical modeling)	0.125	0.954
Wollmann (Physical modeling)	0.102	0.700

After validating the numerical methodology, the continuous casting tundish employed at Gerdau Company was studied. Furthermore, two more cases were performed in order to try to ameliorate the steel quality. Fig. 3 shows the three case studies investigated. For case I, an initial concentration peak is observed. This characterize the existence of short circuiting volume, which promotes the carrying of non-metallic inclusions towards the exit nozzle, and then affecting the final steel quality. In addition, it is observed that once we introduce flow control modifiers, this undesirable volume disappeared. Also, the absence of short circuiting can be visualized as the C-curves present a significant displacement when the three cases are compared, indicating that the first fluid elements are spending more time into the vessel until they leave it.

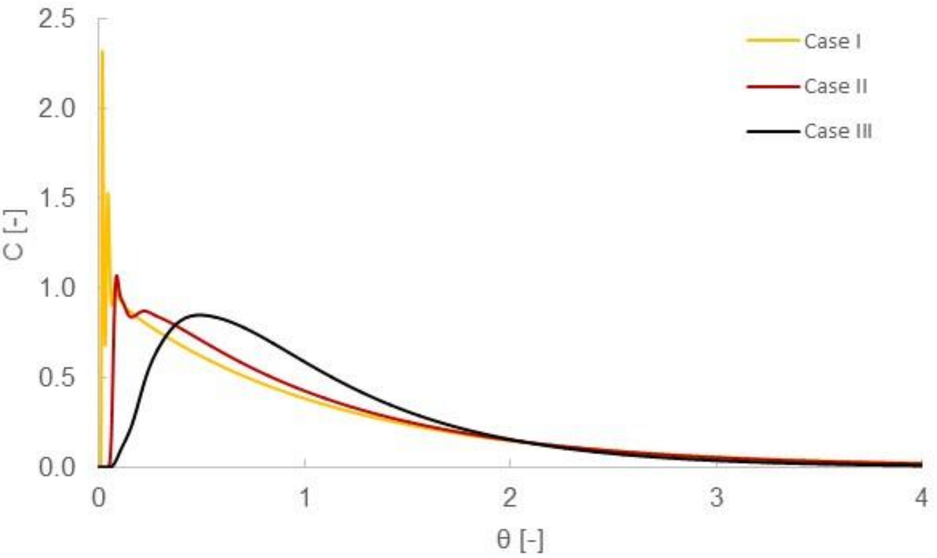


Figure 3: C-curves for different configurations.

Table 3 shows an increase in both minimum and mean residence time for fluid elements into the tundish, which is very important for steel cleanliness once inclusions have more time to float out, and consequently the inclusions can be captured by the slag layer.

Table 3: Minimum and mean residence time for each configuration.

	θ_{min} [-]	θ_{mean} [-]
Case I	0.01	0.95
Case II	0.06	0.99
Case III	0.11	1.04

In order to improve the steel quality, dead volume needs to be minimized, dispersed plug volume maximized, and well-mixed volume be kept in a suitable value. Table 4 shows that these requirements have been achieved with the use of flow modifiers. Hence, their use has been justified.

Table 4: Volume fraction of flow.

	Dead Volume (%)	Dispersed Plug Volume (%)	Well Mixed Volume (%)
Case I	37.7	1.4	60.9
Case II	32.3	7.2	60.5
Case III	18.3	29.9	51.8

CONCLUSIONS

An actual tundish configuration (case I) was analyzed numerically. Also, some geometric modifications (cases II and III) were proposed to the original configuration. With flow modifiers, the minimum and mean residence time were increased. In addition, the characteristic volumes satisfied the required criteria to increase steel quality. Therefore, the use of such flow control devices provides an efficient and reliable way to promote inclusion flotation, and consequently the enhancement of steel cleanliness. Furthermore, comparing the studied cases, we can conclude that the best configuration to be used is the one with dam and weir (case III).

ACKNOWLEDGMENTS

The authors would like to thank the financial support provided by CNPq (The National Counsel of Technological and Scientific Development) and the Gerdau Company for making available to us the actual tundish parameters.

REFERENCES

1. Gardin, P.; Brunet, M.; Domgin, J. F.; Pericleous, K., An experimental and numerical CFD study of turbulence in a tundish container, **Applied Mathematical Modelling**, v.26, p.323-336, 2002.

2. Launder, B. E.; Spalding, D. B., The numerical computation of turbulent flows, **Computer Methods in Applied Mechanics and Engineering**, v.3, p.269-289, 1974.

3. Wilcox, D. C., Reassessment of the scale-determining equation for advanced turbulence models, **AIAA J**., v.26, p.1299-1310, 1988.

4. Kumar, A.; Koria, S. C.; Mazumdar, D., An assessment of fluid flow modelling and residence time distribution phenomena in steelmaking tundish systems, **ISIJ International**, v.44, n.8, p.1334-1341, 2004.

5. Sahai, Y.; Emi, T., Melt flow characterization in continuous casting tundishes, **ISIJ International**, v.36, n.6, p.667-672, 1996.

6. Maliska, C. R., Heat transfer and computational fluid mechanics, 2nd edition., LTC Editora, Rio de Janeiro, 2004. (In Portuguese)

7. User's guide for ANSYS CFX Release 14.0, ANSYS Inc., 2011.

8. Wollmann, A. M., Flow studies in a continuous casting tundish, Thesis. Graduate Program in mining, metallurgy and materials engineering. Federal University of Rio Grande do Sul, Porto Alegre, 1999. (In Portuguese)