APPLICATION OF NUMERICAL OPTIMIZATION TOOLS FOR TUNDISH REFRACTORIES *

Alexandre Dolabella Resende[1]
Rodrigo Nazareth Borges[2]
Rubens Alves Freire[3]

Abstract

The tundish furniture plays a significant role in influencing the flow pattern in the vessel, providing the necessary conditions for clean steel production. The impact pot dissipates the entry jet kinetic energy and directs the flow to the surface, enhancing the molten steel residence time, mixing efficiency and inclusion flotation in the tundish. Moreover, weirs and dams also increase the average flow path towards the strands, further improving the operational conditions for higher quality requirements. Nevertheless, in order to obtain the maximum benefit of steel cleanliness, it is essential that their design is optimum for the specific tundish in consideration. In this study, the application of numerical optimization tools in the design of tundish refractories is presented. The optimizations are based on the results of CFD (Computational Fluid Dynamics) numerical simulations. First, an optimization study of an impact pot design is shown. The impact pot shape is optimized aiming for multiple objectives, being: minimum value of turbulent energy at the slag, minimum value of wall shear stresses at the lining and maximum value of steel residence time. Then, the optimization of the positioning and design of the weir and dam is performed, with the single goal of maximum value for the steel residence time. The results of the optimization studies show that this is a valuable tool to provide better tundish refractory solutions to support clean steel production.

Keywords: Tundish; Numerical Simulation; Optimization; Refractory.

[1] *Mechanical Engineer (UFMG), Development Engineer, Refractory Engineering and Simulations, RHI Magnesita, Contagem, Minas Gerais, Brazil.*
[2] *Mechanical Engineer (UFMG), Marketing Manager, Flow Control Technology, RHI Magnesita, Contagem, Minas Gerais, Brazil.*
[3] *Mechanical Engineer (UFMG), Head of Simulation & PTG West/Americas, Refractory Engineering and Simulations, RHI Magnesita, Contagem, Minas Gerais, Brazil.*

1 INTRODUCTION

The tundish is an important vessel in the continuous casting process, linking the incoming steel from the ladle to the moulds. For many years, the tundish role was limited to the distribution of molten metal to the strands, allowing the continuity of the casting process even between successive ladle changes.

As the demand for quality became more stringent, flow control in the tundish got more attention, as steelmakers worked to make this vessel a place of further refining of the molten steel, instead of only a reservoir. An example of how the fluid flows in the tundish impact on final product quality was given by Guthrie [1] who reported the reject rates of slabs of a particular steel company rise from 0.7% to almost 50% due to an improper placement of an impact pad. The realization of the importance of the fluid flow pattern in the tundish for the final product quality led many researchers and engineers to seek further knowledge in the mathematical modelling of the transport phenomena in the tundish.

The effects of different flow control devices have been thoroughly studied by many researchers [2-8]. What all these studies have in common is that all of them show that the influence of the design and position of these devices in the tundish flow pattern is very significant. Various techniques, such as physical modeling or numerical simulation, have been adopted in order to assess the effectiveness of alternative configurations of tundish furniture and different impact pot geometries in improving the tundish flow pattern.

Although there is a high number of publications about modeling of different tundish flow control devices, the vast majority of them is based in trial and error and manual geometry generation. The parametric optimization of the flow control devices design has not been thoroughly explored and is the main subject of this study.

The impact pot has several important functions in a continuous casting tundish, such as:

- Reducing the splashing during the start-up of the casting.
- Dissipating the kinetic energy of the entry jet from the Ladle.
- Protecting the refractory lining from excessive wear.
- Promoting inclusion flotation.
- Increasing the vessel’s mixing efficiency and minimizing dead zones.

The effectiveness of a given impact pot in complying with these requirements depends largely on its geometric features. Figure 1 shows the RisaImpact™ design features, which includes bottom diffusors and wall tabs to enhance kinetic energy dissipation inside the box, which results in more evenly distributed velocities in the upward flow.

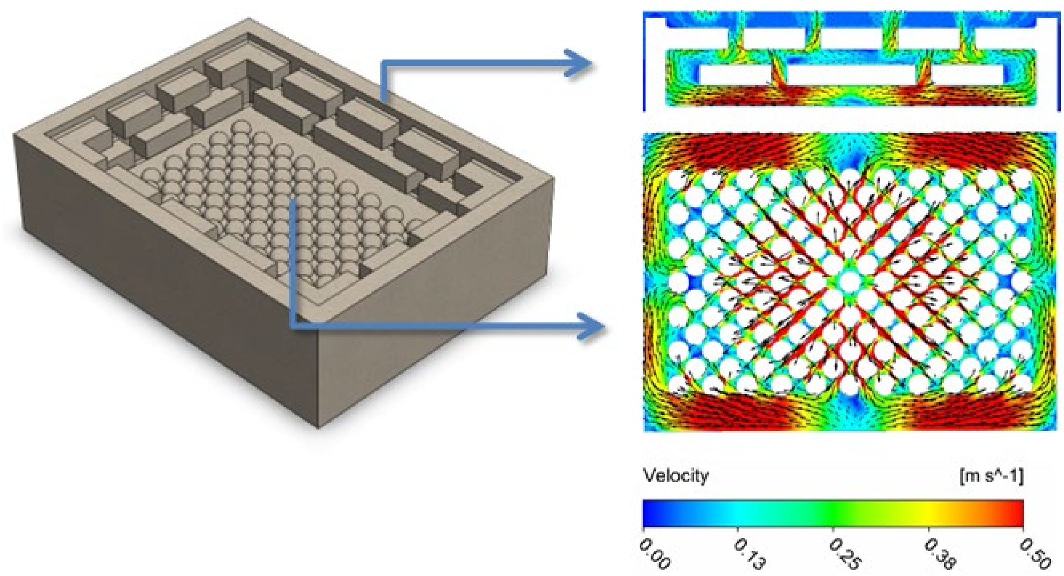

Figure 1: RisaImpact™ design features.

Another example of flow velocity diffusion caused by the impact pot geometry can be seen in Figure 2. The entry jet hits the TUNFLOW Chevron™ bottom with very high velocities (left) and flows upwards with significantly lower velocities due to the barrier effect imposed by the chevrons (right).

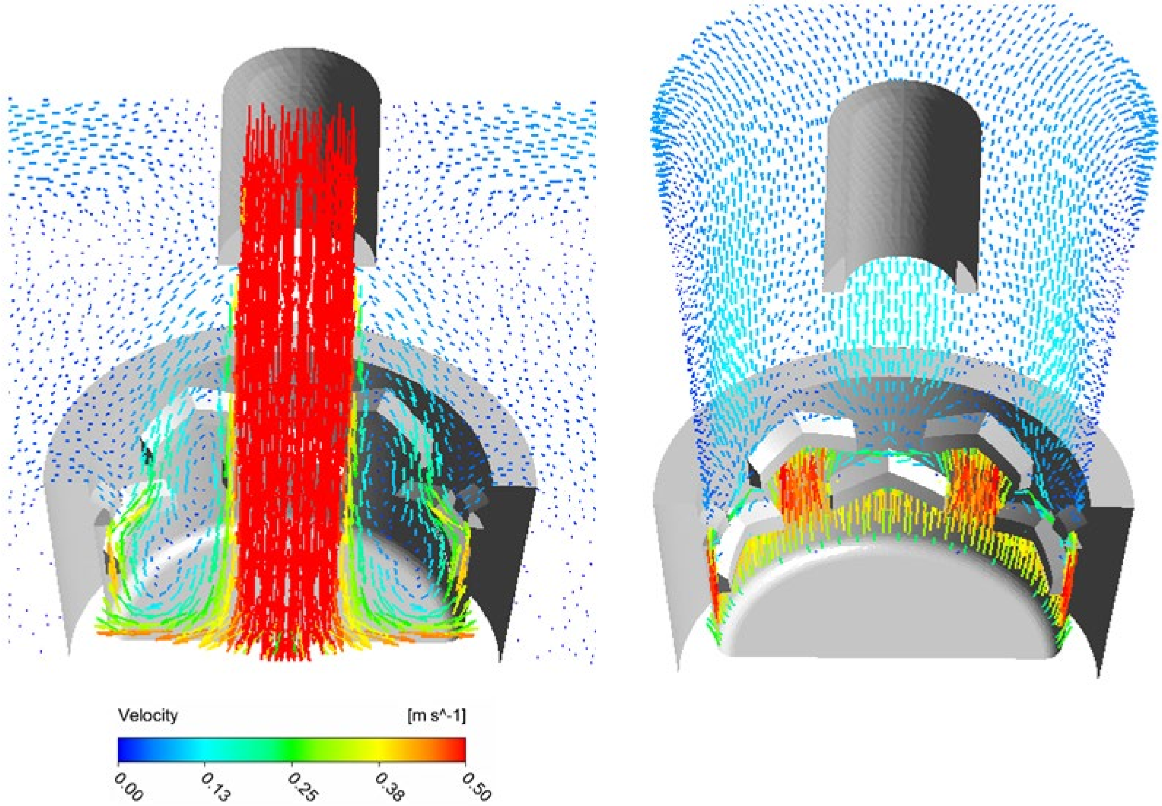

Figure 2: Entry jet's kinetic energy dissipation by a TUNFLOW Chevron™.

As for the weir and dam, some important contributions these components provide to the process are:

- Increasing the shortest flow path to the strands, with consequent increase in residence time and inclusion flotation.
- The weir acts as a barrier to contain slag coming from the ladle in the impact zone, avoiding its flow towards the strand region.
- Dams help to obtain an upward stream, promoting the flotation of inclusions. Dams also help the casting start-up by increasing the bath level upstream during the tundish filling, allowing impurities and loose particles in the lining to float before the strand opens.

Nevertheless, inappropriate configurations of weir and dams can jeopardize the process. Figure 3 shows the flow pattern obtained through a non-optimized configuration of weir and dam. It can be seen that large recirculation zones are formed behind the barriers, which should be related to the positioning and size of these refractory components.

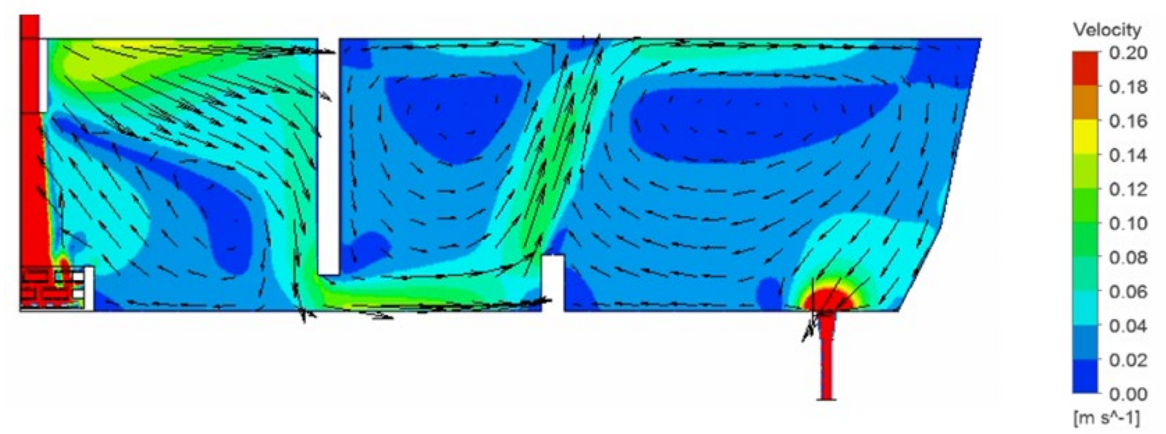

Figure 3: Flow Pattern with large recirculation zones behind the weir and the dam.

2 DEVELOPMENT

2.1 Mathematical Model Description

As it can be seen in both **Figure 1** and **Figure 2**, the geometric features of the impact pot have a significant role in promoting energy dissipation. For this reason, in this study, a parametric optimization of the geometric features of a RisaImpact™ design was performed. The optimized parameters were the vertical distance between the bottom and the first row of wall tabs (P1), the vertical gap between the rows of tabs (P2), the tab length (P3) and the horizontal gap between the tabs (P4). Figure 4 illustrates these parameters.
For the impact pot optimization study, the simulations were performed for a 40-ton two strand tundish with approximately 4 ton/min flow rate. This tundish geometry also has a weir and a dam in each side of the vessel, however their design was kept unchanged for the impact pot optimization study.

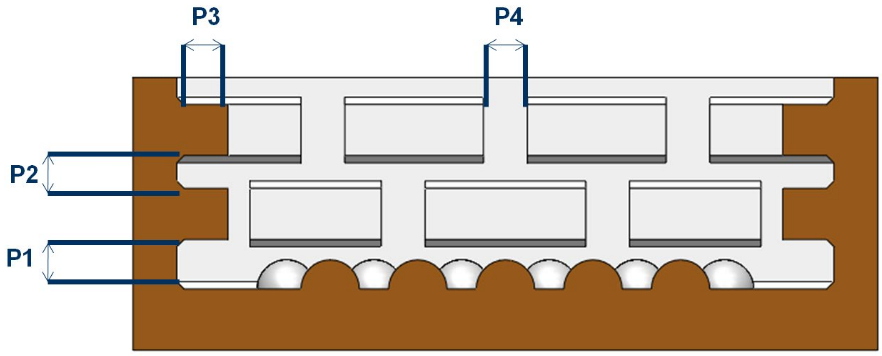

Figure 4: Optimized parameters of the RisaImpact™ geometry.

For the weir and dam optimization, the parameters to consider are the weir submergence depth (P1), the dam height (P2), the distance between the weir and the ladle shroud (P3) and the gap between the weir and the dam (P4). Figure 5 shows these parameters.

The weir and dam optimization study was performed for a different tundish geometry from the impact pot study. In this part of the study, the equipment in consideration is a 70-ton two strand Tundish with a 10 ton/min flow rate. It also had a previous weir and dam configuration, which was regarded as a reference to be compared to the result from the optimization study.

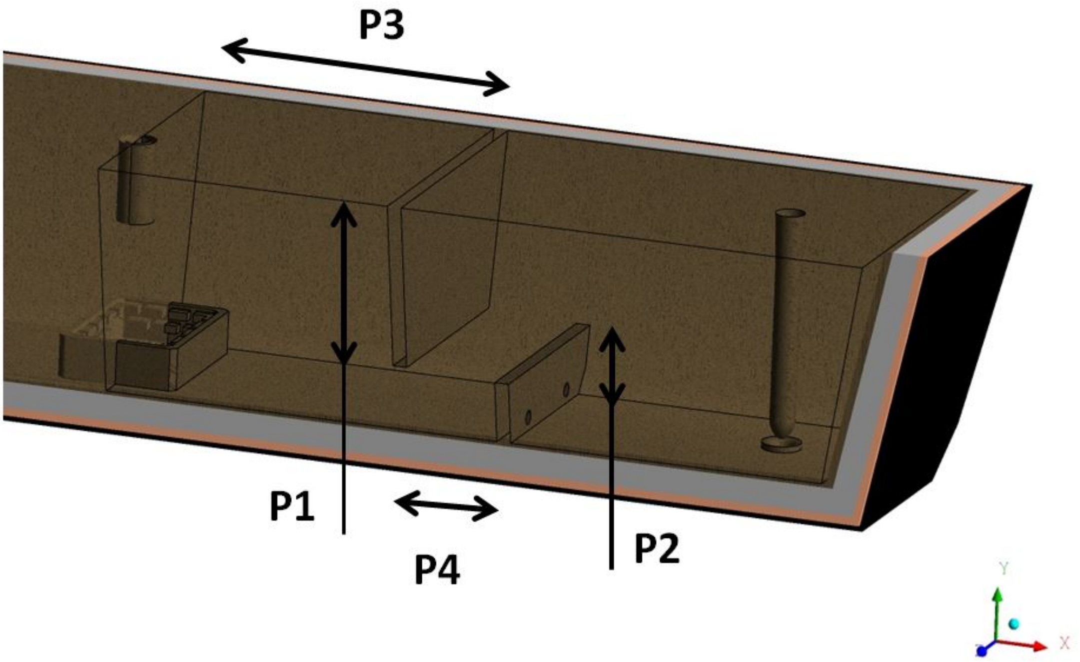

Figure 5: Optimized parameters of the weir and dam configuration.

2.1.1 CFD Model Description [9]

The optimization model is based on the results of CFD simulations. The CFD model solves the Navier-Stokes equations for continuity and momentum (Equation 1 and 2):

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j) = 0 \quad \textbf{(1)}$$

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] + S_M \quad \textbf{(2)}$$

Where ρ is the fluid's density, t is the time, x_j is the coordinate in the j-direction, U_j is the velocity component in the j-direction, P is the pressure field, S_M is the sum of the body forces and μ_{eff} is the effective viscosity accounting for turbulence, given by Equation 3:

$$\mu_{eff} = \mu + C_\mu \rho \frac{k^2}{\varepsilon} \quad \textbf{(3)}$$

Where μ is the fluid's molecular viscosity, C_μ is a constant, k is the turbulent kinetic energy and ε is the dissipation rate of turbulence.
Equations (4) and (5) represent the transport equations for turbulent kinetic energy and dissipation rate of turbulence:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon \quad \textbf{(4)}$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_i \varepsilon) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}(C_{\varepsilon 1} P_k - C_{\varepsilon 2}\rho\varepsilon) \quad \textbf{(5)}$$

Once the flow field is calculated, a Residence Time Distribution analysis is performed to characterize the flow according to the definitions of Plug Volume, Dead Volume and Mix Volume. The definitions adopted in this work are from Sahai [10]. To perform the RTD study, a numerical simulation of a tracer transport in the calculated flow field was performed. The transport equation for the tracer is given by Equation 6:

$$\frac{\partial(\rho\varphi)}{\partial t} + \nabla\cdot(\rho \boldsymbol{U}\varphi) = \nabla\cdot\left(\left(\rho D_\phi + \frac{\mu_t}{Sc_t}\right)\nabla\varphi\right) + S_\varphi \quad \textbf{(6)}$$

Where φ is the tracer concentration, Sc_t is the turbulent Schmidt number, S_φ is a source term for the concentration, μ_t is the eddy viscosity and Dϕ is the kinematic diffusivity of the tracer.

2.1.2 Optimization Model Description

The optimization model is divided in 3 steps:

1. First, a Design of Experiments (DOE) matrix is generated with several different combinations of the design parameters (P1, P2, P3 and P4) in order to cover the whole range of admissible parameters with the fewest number of simulations as possible.
2. Then, a Response Surface is adjusted in the obtained data from the CFD model and extra refinement points (represented as additional combinations of

the design parameters) are computed and added to the response surface calculation in order to increase its accuracy.

3. Finally, a verification algorithm is run with the goal of assessing the accuracy of the response surface and searching for the global maximum and minimum of the desired outputs.

In this study, for the impact pot optimization, the desired results from the optimization model were the global minimum for the turbulence kinetic energy at the slag surface, the global minimum for the wall shear stresses in the refractory lining and the global maximum for the minimum residence time. As it consists in a multiple objective problem, the optimization model solution consisted of three alternative candidates, each with a higher score in one of these goals.
For the weir and dam optimization, only the global maximum of the minimum residence time was defined as a desired output from the model. Therefore, the solution consists of a single combination of design parameters.

2.2 Results

2.2.1 Impact Pot Optimization Results

Figure 6 shows the obtained results for the three candidates obtained through the optimization model for the impact pot. The central black square with the thicker lines is the reference case, scoring 0% at every desired output. The results for the other cases have been calculated in relative percentage compared to the reference case. In the results where improvement has been obtained, positive percentage values have been assigned, even if the improvement means a reduction in the absolute value of the variable (e.g. if the turbulence value reduces in 30%, this result is shown as positive 30% in Figure 6 because it means a 30% improvement has been obtained with respect to this variable).
Three candidates have been obtained through the optimization calculations. Candidate 01 scored the highest performance in minimizing wall shear stresses, showing a reduction of 22% compared to the reference case. Figure 7 shows the contours of wall shear stresses comparing the reference case with the best configuration for minimum wall shear obtained through the optimization.
Candidate 02 was proposed by its higher score in maximizing the minimum residence time. This is translated into higher values of Plug Volume and lower values for the Dead Volume. The results in Figure 6 are shown with respect to the obtained Plug Volume and Dead Volume results rather than the raw results for the minimum residence time due to the correlations between the residence time distribution (RTD) results and different indicators of process performance. Higher values of Plug Volume are associated with better conditions for inclusion flotation while lower values of Dead Volume are associated with increased mixing efficiency in the vessel, which avoids problems such as cold zone development and short-circuiting.
Candidate 02 showed a 3% increase in the Plug Volume result and a 2% decrease in the Dead Volume result. The obtained curves are shown in Figure 8. Although it is an improvement, the results are very modest. The reason for the not so significant improvement in the RTD results is that, for the specific tundish in consideration, the already existing weir and dam configuration have a strong influence in the flow pattern, limiting the magnitude of improvement that can be obtained by changes in the impact pot design. In future studies, the optimization model for the impact pot

design should be applied to a tundish geometry without weirs and dams in order to assess if more significant improvements in the RTD results can be obtained.

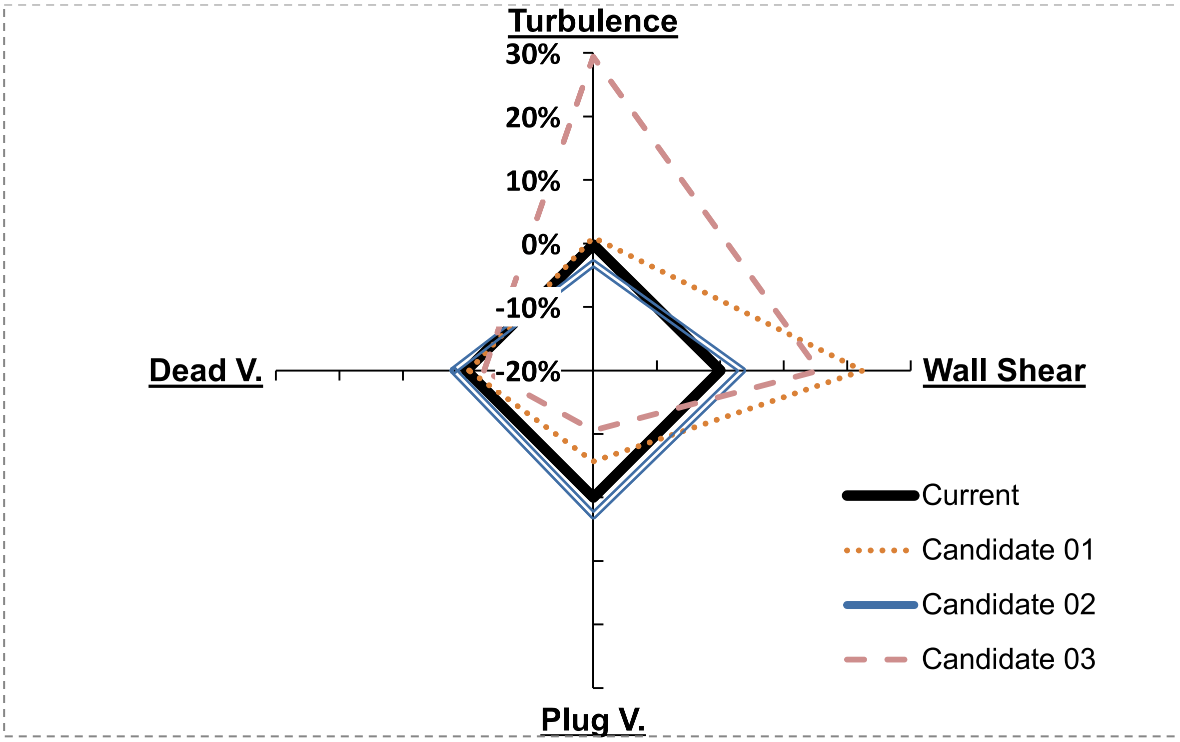

Figure 6: Results of the optimization model for the impact pot.

The results for the optimization Candidate 03, which showed the highest score for reduction of the turbulence kinetic energy at the slag, are shown in Figure 9. The turbulence kinetic energy has been reduced by 29% in the optimized configuration compared to the reference case.

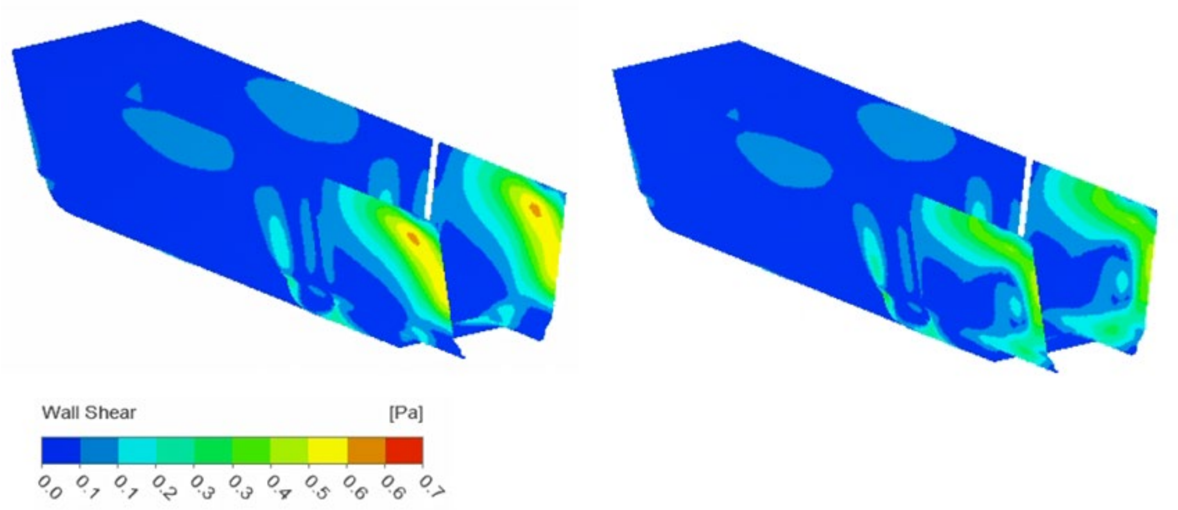

Figure 7: Comparison between wall shear stresses of the reference case (left) and Candidate 01 proposed by the optimization model (right).

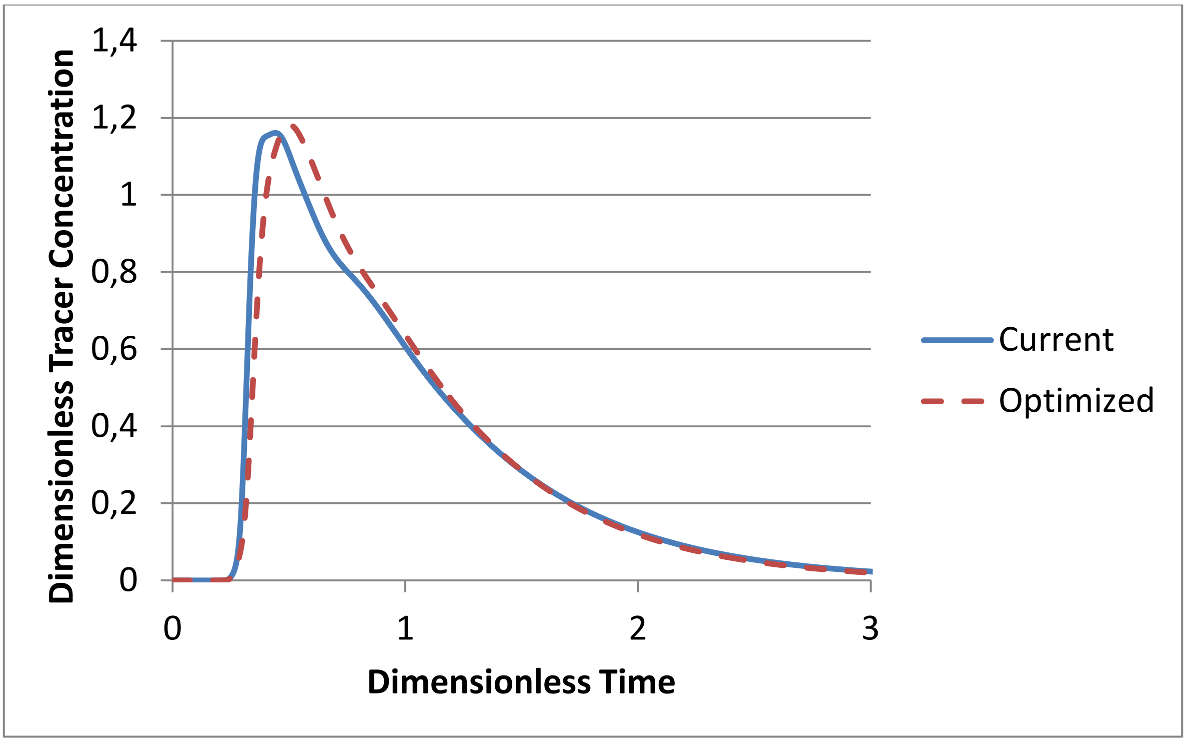

Figure 8: Comparison between the obtained RTD curves for the current case and Candidate 02 proposed by the optimization.

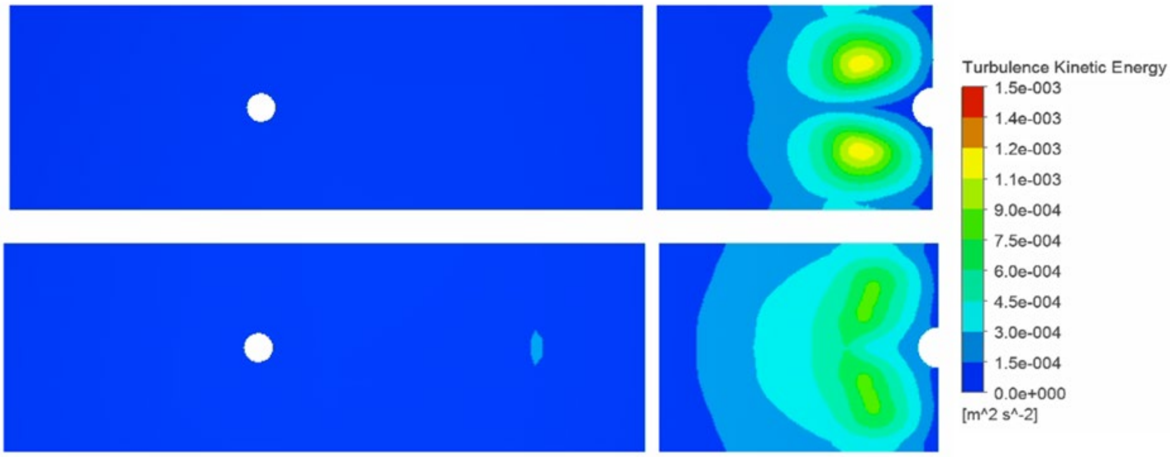

Figure 9: Comparison between the turbulence kinetic energy at the slag surface for the reference case and Candidate 03 proposed by the optimization.

2.2.2 Weir and Dam Optimization Results

Regarding the optimization study for the weir and dam positioning and design, Figure 10 shows the velocity contours and vectors comparing the obtained flow patterns in the reference configuration against the optimized configuration. It can be seen that the large recirculation zones behind the barriers have been significantly reduced in the optimized configuration. Moreover, the average flow velocities have also been reduced, which translates into more favorable conditions for inclusion flotation.

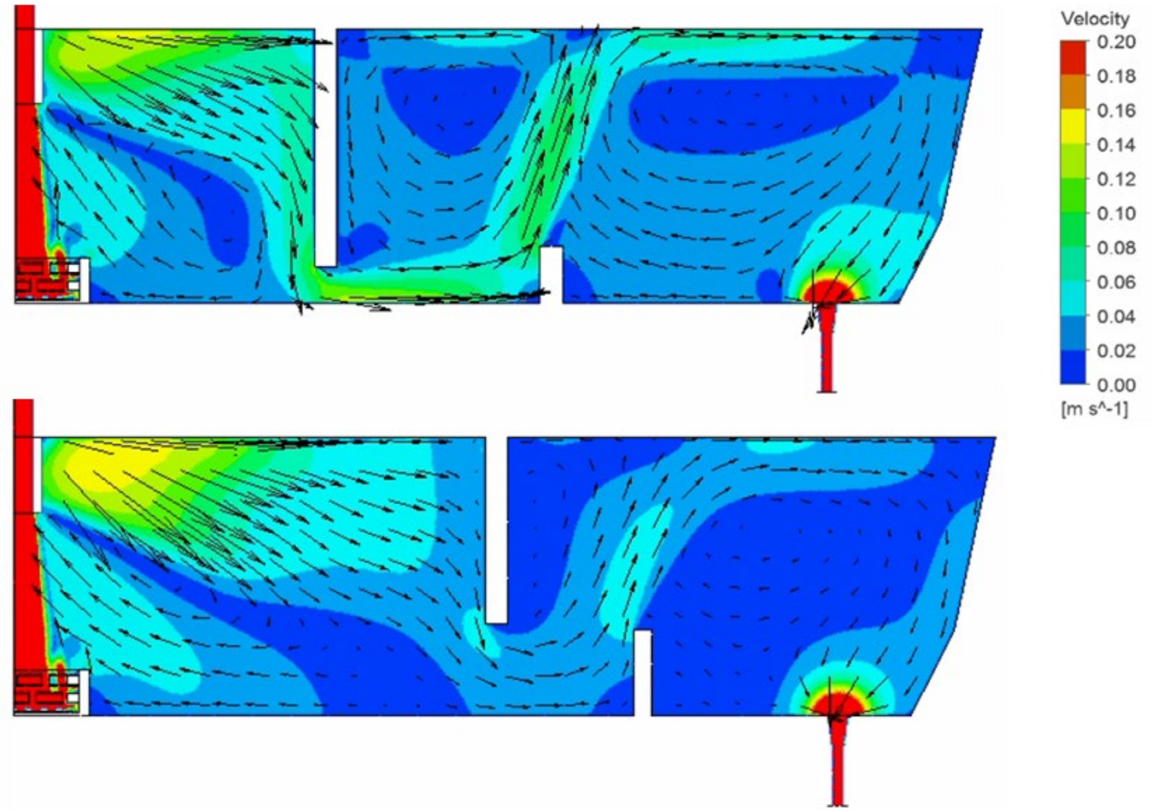

Figure 10: Comparison between the obtained flow patterns for the reference case (above) and optimized weir and dam (below).

Figure 11 shows the obtained RTD curves for both cases. Differently from the impact pot optimization study, for this case the RTD results have been significantly improved.

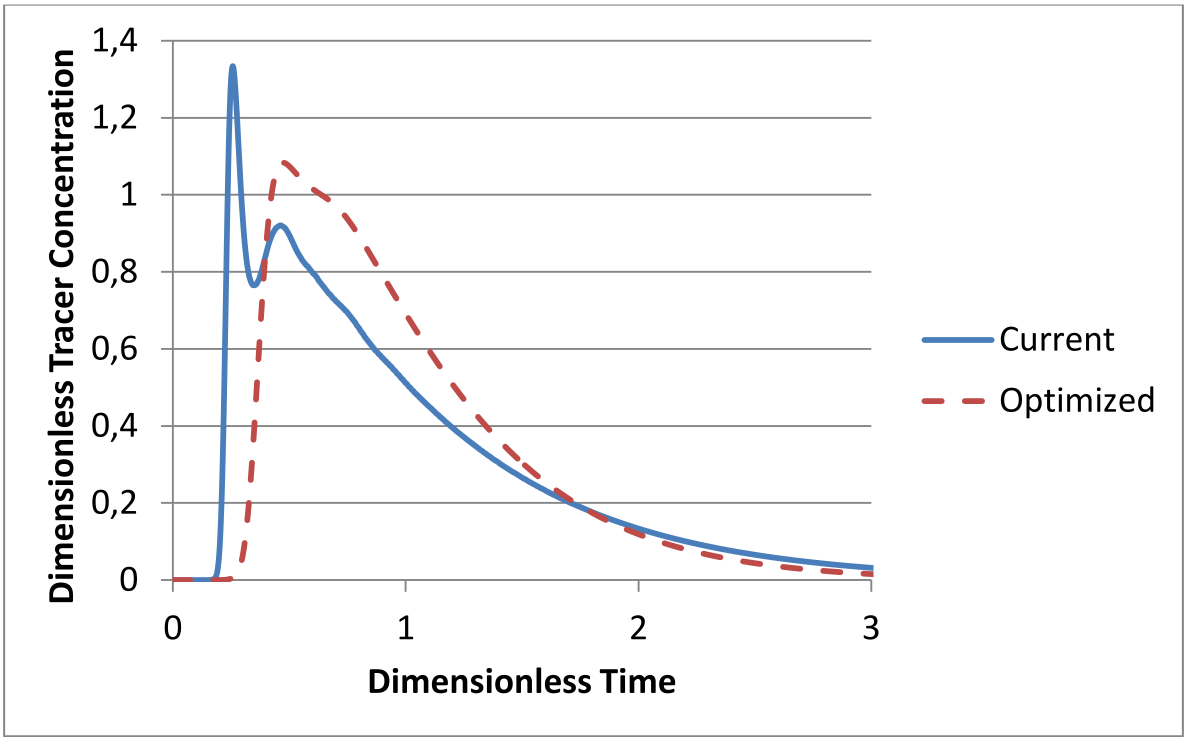

Figure 11: Comparison between the obtained RTD Curves for the reference case and the optimized weir and dam configuration.

The Plug Volume fraction increased 42% in the optimized case compared to the reference, the minimum residence time increased 54% and the Dead Volume fraction reduced 44% compared to the reference case. The improvement that has been obtained by optimizing the weir and dam configuration is very significant and it means achieving better process performance without adding significant costs, as the improvement was obtained through redesigning and repositioning the weir and dam, not by adding extra components to the tundish furniture.

2.2.3 Results Discussion

As discussed in the introduction section of this work, several authors have studied the influence of flow control devices on the tundish flow pattern. There is a clear agreement between all these studies with respect to the conclusion that the position and design of the flow control devices have a significant influence in the quality parameters of the tundish flow.
The results of the present work also agree with this conclusion. Significant changes have been observed as the design of the impact pot and weir and dam arrangement were modified. The only result which did not show significant change were the RTD results in the impact pot optimization, however this can be attributed to the fact that the already existing weir and dam configuration in the tundish contributed to attenuate the effects of changes in the impact pot design. Therefore, only the results close to the impact region were significantly influenced by the changes in the impact pot geometry, such as the turbulence in the slag surface and the wall shear stresses in the refractory lining. This should be significantly different in case the optimization study for the impact pot is performed with respect to a tundish without other flow modifiers. It is expected that, in this case, the RTD results should also change significantly with changes in the impact pot design. In future work, the optimization algorithm for the impact pot should be applied to a tundish without other flow modifiers.
Regarding the improvements obtained in the presented cases in this work, it can be seen that the application of numerical optimization tools is a very efficient method to improve the quality parameters of the tundish flow. In most studies in the literature, improvements in the range of 40-50% are only obtained by adding extra components in the Tundish furniture. However, by applying optimization methods, such improvements can be obtained by simply redesigning and repositioning the already existing refractory components. Therefore, the presented technique is a valuable tool to add value to the process without adding costs.

3 CONCLUSION

- The impact pot design and the weir and dam arrangement have a significant influence in the quality parameters of the tundish flow pattern.
- It is possible to significantly improve the tundish performance by carefully designing these refractory components.
- Numerical optimization tools are a valuable technique to improve the design of the tundish flow control devices.

REFERENCES

1 Guthrie RIL. Fluid Flows in Metallurgy – Friend or foe?. Metallurgical and Materials Transactions B. 2004; Volume 35B: 417-437.

2 Chattopadhyay K, Isac M, Guthrie RIL. Effect of flow modifiers on liquid metal cleanliness in four-strand delta shaped billet caster tundish. Ironmaking and Steelmaking. 2012; Volume 39: 454-462.

3 Merder T. The Influence of the Shape of Turbulence Inhibitors on the Hydrodynamic Conditions Occurring in a Tundish. Archives of Metallurgy and Materials. 2013; Volume 58: 1111-1117.

4 Re Z, Zhou K, Liu S, Xiong W, Li B. Numerical Modeling of the Fluid Flow in Continuous Casting Tundish with Different Control Devices. Abstract and Applied Analysis. 2013; Volume 2013: 1-8.

5 Fan CM, Shie RJ, Hwang WS. Studies by mathematical and physical modelling of fluid flow and inclusion removal phenomena in slab tundish for casting stainless steel using various flow control device designs. Ironmaking and Steelmaking. 2003; Volume 30: 341-347.

6 Liu J, Yan H, Liu L, Wang X. Water Modeling of Optimizing Tundish Flow Field. Journal of Iron and Steel Research, International. 2007; Volume 14: 13-19.

7 Zheng S, Zhu M. Optimization of flow control devices in a ten-strand billet caster tundish. China Foundry Research & Development. 2016; Volume 13: 414-421.

8 Espino-Zárate A, Morales RD, Nájera-Bastida A, Macías-Hernández MJ, Sandoval-Ramos A. Fluid Flow and Mechanisms of Momentum Transfer in a Six-Strand Tundish. Metallurgical and Materials Transactions B. 2010; Volume 41B: 962-975.

9 ANSYS 18.1. CFX Theory Guide. Release 18.1. 2017.

10 Sahai Y, Emi T. Melt Flow Characterization in Continuous Casting Tundishes. ISIJ International. 1996; Volume 36; 667-672.