J. CUPEK, M. TKADLEČKOVÁ, T. MERDER, J. WALEK, M. SATERNUS, J. PIEPRZYCA
ISSN 0543-5846
METABK 62(3-4) 335-338 (2023)
UDC – UDK 669.18:669.186:532.5:518.1=111

COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS OF MEDIUM FLOW AND REMOVAL OF INCLUSIONS IN A TWO-STRAND TUNDISH

Received – Primljeno: 2022-10-17
Accepted – Prihvaćeno: 2023-02-10
Original Scientific Paper – Izvorni znanstveni rad

The article presents the results of the CFD simulation of the method of medium flow and mixing, as well as the movement and removal of solid particles in the model of two-strand tundish as a result of installed internal arrangement. Two variants including turbulence inhibitor and impact pad with a ball cutting area were modeled for the tests. CFD simulations are a continuation of previous tests with the use of the physical water model of the continuous steel casting (CSC) device.

Keywords: steel, tundish, liquid flow, turbulence inhibitor, impact pad, numerical modeling

INTRODUCTION

Metallurgical cleanliness of steel products is one of the basic criterion for assessing their quality. The flow and mixing method of liquid steel in tundish has a significant impact on the effectiveness of removing impurities.

Tundish is the last place during the continuous steel casting, in which the amount of non -metallic inclusions (NMIs) in steel can be reduced or modified and the steel is homogenized [1, 2]. The steel flow in tundish is optimized by its internal system, in which can be installed flow regulators such as turbulence inhibitors (TI), impact pads (IP), baffles, dams and weirs or another argon stirring element placed in the bottom of the tundish. Most commonly used are turbulence inhibitor or impact pad, therefore this particular research was focused on optimizing the steel flow in the two-strand tundish using this elements of arrangement [3].

Turbulence inhibitor or impact pad prevent spattering of steel after direct contact of the steel with the bottom of the ladle. They significantly reduce the turbulent flow of steel and eliminate the erosion of the lining of the bottom of the tundish. They direct the steel flow back into the upper part of the surface, creating an area in which the steel flows in plug flow and is favorable for the flotation of NMIs [4].

Due to the importance of this problem, research is carried out in many centers in the world in order to obtain the required structure of liquid steel flow through the tundish. Model tests are the most commonly used tool for solving this problem. The theoretical foundations of this research are fluid mechanics [5]. Based on the Navier-Stokes equation, physical and CFD models are created for testing specific solutions of the tundish arrangement. During the research of liquid steel motion in a tundish, the size of the characteristic flow zones (turbulent flow, plug flow and dead flow zone) is observed as a result of the flow regulators used.

In this way, the optimal tundish arrangement is determined from the point of view of the expected effects of liquid steel refinement. The previous article [6] presents the results of the research using the tundish water model. CFD numerical simulations are a continuation of these studies. ANSYS Fluent software was used for calculations [7].

NUMERICAL MODELING

The assumption of CFD numerical simulations was that the obtained results were used to combine them with the results of tests carried out using the physical model of the Continuous Steel Casting (CSC) device. Therefore, liquid steel was replaced with water for numerical analysis, while non-metallic inclusions (NMIs) with microparticles [8]. Numerical simulations of the water flow and distribution of model NMIs (microparticles) were carried out using commercial Ansys Fluent software ver. 19.2 [7].

The work area of the object (tundish model) was made in Design Modeller, Figure 1. Tundish is characterized by a 0,025 m bottom in the area of outlets. In the case of the geometry performed, the computing mesh was compacted in the inlet and outlet area. Computational density analysis has shown that a mesh containing 520 000 cells will be sufficient. The quality of calculated mesh was checked using a criterial skew angle [7].

J. Cupek, J. Walek, M. Tkadlečková: VSB - Technical University of Ostrava, Ostrava, Czech Republic
J. Pieprzyca, T. Merder, M. Saternus: Silesian University of Technology, Faculty of Material Science, Katowice, Poland
Corresponding author: jacek.pieprzyca@polsl.pl.

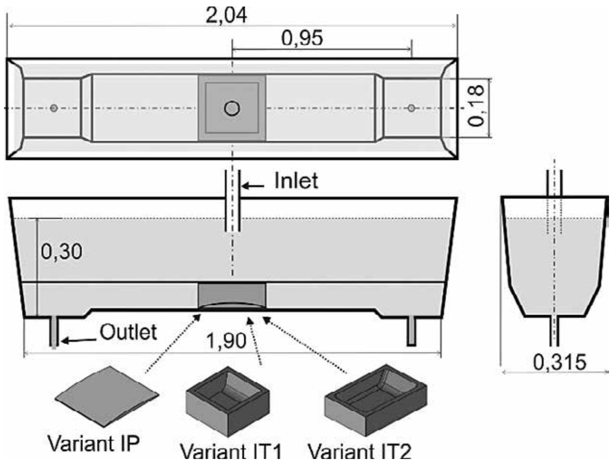

Figure 1 The geometric dimensions / m of the tundish

The extended characteristics of the applied mathematical model can be found in work [9]. The k-ε model was used to describe the turbulent steel flow field [7].

On the edge corresponding to the tundish inlet, the boundary condition was implemented based on the conditions presented in Table 1. The intensity of the outflow from tundish is calculated from the mass balance. Operational and boundary conditions (see Table 1) of the mathematical model correspond to the conditions of research on the physical (water) model [10].

Table 1 **Physical quantities accepted for model tests**

Parameter	Unit	Value
Inlet velocity	$m \cdot s^{-1}$	1,31
Inlet turbulence intensity	%	10
Water density	kg/m^3	998,2
Water kinematic viscosity	m^2/s	$1{,}00 \cdot 10^{-6}$
Microparticles density	kg/m^3	120
Liquid steel density	kg/m^3	»7000
Liquid steel kinematic viscosity	m^2/s	$0{,}913 \cdot 10^{-6}$
Al_2O_3 inclusions density	kg/m^3	3900
Relation of density of inclusions and liquid steel	-	0,5571
Relation of density of microparticles and water	-	0,12

The SIMPLEC algorithm was used in numerical simulations. During iteration, the convergence was assumed to reach a point where all the normalized residuals were smaller than 10^{-6}. Calculations were made on a computing server in Second Order Upwind. Model NMIs (microparticles) trajectories were calculated using the Lagrange method. This method consists of solving equation for transport of solid particles movement in a previously defined vector medium (water) speed field, taking into account the additional effects resulting from turbulent flow. To this end, it was necessary to supplement the system of equations with differential equation describing the movement of particles in the liquid phase (Discrete Random Walk – DRW) [7].

The calculations assumed that modeled solid particles (microparticles) are spherical and are introduced through inlet to tundish with a constant initial speed, equal to the speed of the medium. On the surface of free metal, solid particles are captured, which corresponds to the ideal conditions of NMIs absorption. On the other surfaces of tundish (bottom and walls), the particles are reflected, after which they leave the tundish outlets with the bath.

In order to ensure a even distribution of model NMIs, an introduction of a total of 5 200 solid particles was assumed on the inlet surface. In CFD simulations, the following groups of solid particle diameter were adopted: 5, 25, 35, 45, 50 µm.

According to the mathematical analysis described in the work [10], the solid particles analyzed in the research correspond in industrial conditions to the diameters of NMIs as depending:

$$d_{NMI} = 2 \cdot d_{par.} \qquad (1)$$

where: d_{NMI} – diameter of spherical (globular) NMI in steel, $d_{par.}$ – diameter of a spherical model microparticle.

Simulations were carried out for three different flow control devices (IP, IT1, IT2) installed in the tundish working space (see Figure 1). They were described in detail at work [6].

RESULTS AND DISCUSSION

In order to evaluate the obtained results of the CFD simulation, the object was cut with a vertical transverse plane passing through the inelet and outlets tundish axis. The results of the CFD calculations, illustrating the expected movement of the medium in the tundish for three different arrangement variants, are shown in Figure 2.

All (see Figure 2) tested configurations had different flow characteristics. However, significant differences were observed between the IP variant and the IT1 and IT2 variants. In the IP variant, a large dispersion of the velocity vectors was observed in the inflow zone due to the reflection of the stream from the impact pad in a diagonal direction to the steel mirror. This results in a clear expansion of the stream in a large volume of the model liquid and the formation of a plug flow front.

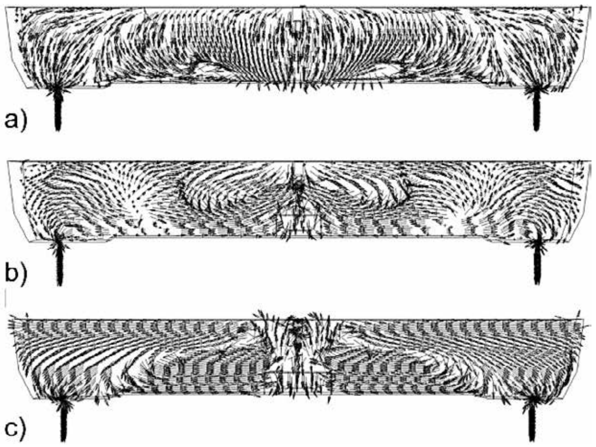

Figure 2 Distributions of the water velocity vectors - variant: a) IP, b) IT1, c) IT2

This is a favorable phenomenon from the point of view of obtaining the required ratio of turbulent to plug flow. This favors the outflow of NMIs to the slag and reduces the possibility of secondary contamination of steel at the metal-slag interface. This is especially true for large NMIs for which buoyancy is the dominant carrier. For variants IT1 and IT2, the simulant flow is different. In both cases, a strong reflection of the model liquid stream and the direction of its velocity vectors into a small surface of the steel mirror is observed. As expected and the role of the turbulence inhibitor, the turbulent flow zone is significantly limited. In the IT2 variant, a particularly strong rise of the model liquid stream was observed, which adversely affects the state of the metal-slag interface and may be the cause of excessive waving of the steel mirror. The proportions between the turbulent and plug flow zones are clearly unfavorable for the former. In addition, in the IT2 variant, there is a tendency to the occurrence of a circulating flow, which causes horizontal stratification of the flow of the model liquid, visible in the figure in the form of a clear boundary of the flow velocity of the model liquid towards the outflows in the zone under the slag. This phenomenon hinders the free flow of NMIs into the slag from the zone below this boundary. However, it may positively affect on the absorption of NMIs of very small sizes by the slag, carried by the energy of the flow of the model liquid.

Figure 3 shows the distribution of the kinetic energy of water turbulence on the control plane.

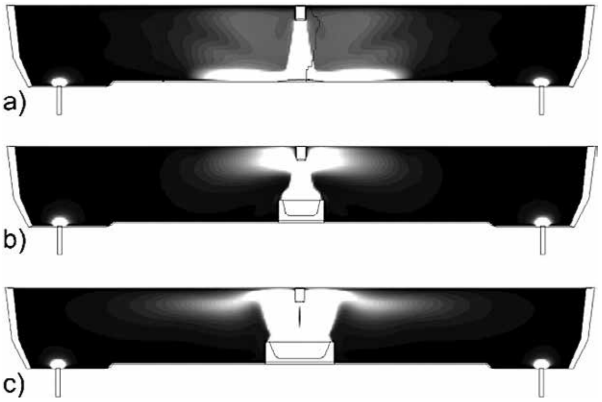

Figure 3 Isolines of liquid turbulence kinetic energy - variant: a) IP, b) IT1, c) IT2

As in the case of velocity vector analysis, significant differences were observed between the IP variant and the IT1 and IT2 variants in the case of the determined turbulence fields. In the case of the IP variant, the turbulence field is located in the zone of impact of the inlet liquid stream against the impact pad, while in the case of IT1 and IT2 variants, this field is located in the upper part of the inlet zone under the steel mirror. A significant reduction in the range of the turbulent flow zone in variants IT1 and IT2 is clearly visible. This is not conducive to the required homogenization of the liquid steel and hinders the outflow of NMIs, especially of large sizes.

The determined velocity vectors and turbulence fields were confirmed by the numerical simulation of the removal of solid particles in the tested reactor, Figure 4. It can be stated that differences were also obtained when evaluating the effectiveness of removing non-metallic inclusions. The most noticeable differences were achieved for the smallest particles with a size of 5 μm, where for the IT2 variant more than 80 % of the introduced non-metallic particles were removed. By contrast, less than 70 % was removed for the IP variant. For the largest particles with a diameter of 50 μm, the IP variant performed best, removing over 91 % of the particles.

The obtained results were supplemented with RTD (residence time distribution) F-type curves - Figure 5.

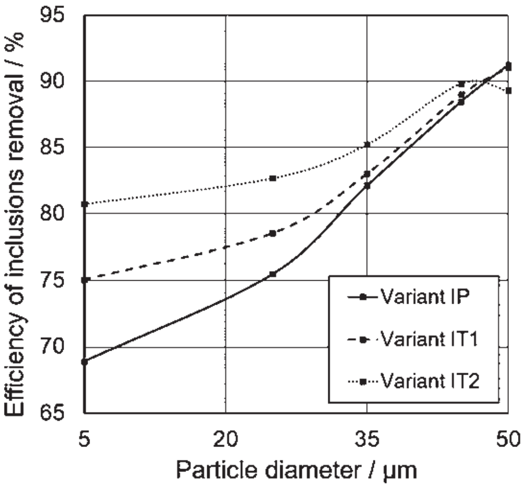

Figure 4 The amount of particles that were captured by the free surface for the tested variants

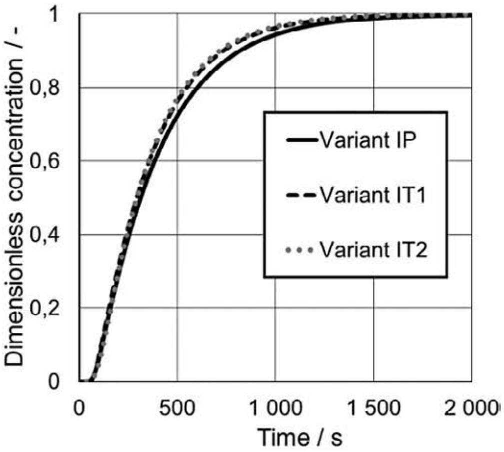

Figure 5 RTD curves (F-type) for the tested variants

When analyzing the determined RTD curves, it should be noted that their character is similar for all variants. No differences were found in variants IT1 and IT2. In the IP variant, a certain slowdown of the mixing was found in the range of 0,6 – 0,9 dimensionless tracer concentration, which may favor the free flow of non-metallic inclusions into the slag.

CONCLUSIONS

The results obtained during the research can be summarized as follows:

- The nature of the model liquid flow in the IP variant is significantly different from the flow in the IT1 and IT2 variants. It found a much better ratio of the size of the turbulent flow zones to the plug flow zones. Such a flow is conducive to limiting the growth of dead zones and is beneficial from the point of view of the possibility of removing NMIs. This is especially true for larger inclusions.
- In variant IT1, a limited range of the turbulent flow zone located in the inlet region of the model liquid was found. The ratio of this range to the plug flow range is unfavorable. It favors the tendency to create harmful dead zones, what does not guarantee successful removal of NMIs.
- In the IT2 variant, as a result of the strong directing of the stream of the model liquid towards steel mirror, a slight increase in the turbulent flow zone occurs in relation to the IT1 variant. However, the risk of occurrence of unfavorable phenomena identified in variant T1 does not change. In addition, a tendency to excessive waviness of the steel mirror was found. On the other hand, the presence of a recirculating flow is beneficial, which may favor the removal of small NMIs.

Acknowledgements:

This work was part of Doctoral grant competition VŠB-TU Ostrava No CZ.02.2.69/0.0/0.0/19_073/0016945, part of DGS/TEAM/2021-002, Czech Republic. The work was created as part of Student Grant Competition No SP2022/15, SP2022/68, SP2022/83, SP2022/84. The work was part of the project No. CZ.02.1.01/0.0/0.0/17_049/0008399, EU and Czech financial funds.

The article was supported by Statutory Research 11/020/BK_23/0104 (BK-219/RM2/2023) of Silesian University of Technology.

REFERENCES

[1] S. Louhenkilpi. Continuous casting of steel. treatise on process metallurgy vol. 3: Industrial Processes, Publishing of the Royal Institute of Technology, Stockholm, Sweden 2014, 373-434.

[2] W.R. Irwing. Continuous casting of steel, Publishing of the Institute of Materials, London 1993.

[3] Y. Sahai, Tundish technology for casting clean steel: A review. Metall. Mater. Trans. B 47 (2016) 4, 2095-2106.

[4] B. Bulko, I. Priesol, P. Demeter, P. Gasparovic, D. Baricowa, M. Hrubovcakowa. Geometric modification of the tundish impact point, Metals (2018) 8, 1-11.

[5] K. Jeżowiecka, H. Szewczyk. Fluid mechanics, Publishing House of the Wroclaw University of Technology, Wroclaw 2001.

[6] J. Cupek, J. Pieprzyca, J. Walek, M. Saternus, M. Tkadlečková, T. Merder. Optimization of the steel flow in the two-strand tundish using different geometry of impact pad, Metalurgija 62 (2023) 2, 179-182.

[7] ANSYS Fluent ver. 19.2 – user's guide, Canonsburg, USA 2022.

[8] Information Materials, 3M_Glass-Bubbles, 2019.

[9] M. Tkadleckova, J. Walek, K. Michalek, T. Huczala. Numerical analysis of RTD curves and inclusions removal in a multi-strand asymmetric tundish with different configuration of impact pad, Metals (2020) 10, 1-20.

[10] T. Merder, M. Warzecha, P. Warzecha, J. Pieprzyca, A. Hutny. Modeling research technique of nonmetallic inclusions distribution in liquid steel during its flow through the tundish water model, Steel Res. Int. 90 (2019) 7, 1-10.

Note: The responsible for English language is Paulina Pieprzyca