T. MERDER*[#]

NUMERICAL ANALYSIS OF THE STRUCTURE OF LIQUID FLOW IN THE TUNDISH WITH PHYSICAL MODEL VERIFICATION

Detailed studies of the movement of liquid steel (hydrodynamics) on a real object are practically impossible. The solution to this problem are physical modelling carried out on water models and numerical modelling using appropriate programs. The method of numerical modelling thanks to the considerable computing power of modern computers gives the possibility of solving very complex problems.

The paper presents the results of model tests of liquid flow through tundish. The examined object was model of the two-nozzle tundish model. The ANSYS Fluent program was used to describe the behavior of liquid in the working area of the tundish model. Numerical simulations were carried out using two numerical methods of turbulence description: RANS (Reynolds-Averaged Navier-Stokes) – model k-ε and LES (Large Eddy Simulation). The results obtained from CFD calculations were compared with the results obtained using the water model.

Keywords: steelmaking, continuous casting, tundish, numerical modelling, physical modelling

1. Introduction

Continuous casting (CC) of steel is currently the most commonly used process for the production of steel semi-finished products. Therefore, research related to the optimization and improvement of the steel casting process is a key to modernizing existing and implementing new solutions to industrial practice. Although this method has been used on a large scale in the steel industry for several decades, there are still problems with the quality of continuous ingots.

The tundish – regardless of the passage of time – is and will remain an essential component of every CC equipment. Today's tundish is identified as a high-temperature flow chemical reactor in which both autonomous and coupled processes are realized, the effects of which affect the metallurgical purity level of cast steel and consequently the final steel product.

Analysis of the liquid steel flow structure, which is very important when considering the obtaining of high-quality ingots, is not possible under industrial conditions. It is possible to perform partial measurements on a real object, however they do not give a full picture about the analysed issue. In addition, these research are very often difficult and expensive.

Modelling research, on the other hand, are a commonly used research method enabling to reveal the nature of liquid steel flow in tundish and the influence of technological factors on the hydrodynamic conditions.

Modelling research include experiments using water models (physical modelling) and numerical simulations (CFD – Computational Fluid Dynamics) [1-8].

Numerical modelling is considered to be an alternative method for physical modelling, which should be more closely associated with experimental research. In addition, it was assumed that both these methods are combined in one research program because they support and complement each other [5-8]. These methods together create the so-called hybrid modelling. This type of modeling is often verified by partial determinations carried out under industrial conditions. This is favored by the dynamic development of computer technology (constantly increasing computing power of computers) and the emergence of more and more perfect numerical procedures.

Systematic development of computational methods of flow modeling has resulted in the availability of appropriate tools to study phenomena in the field of hydrodynamics and mass and heat exchange occurring in steel aggregates. The mentioned tools are commercial computer programs in the field of CFD. The complex issue of the flow and mixing process of liquid steel in a tundish can be expressed by means of forces balance in the element of flowing steel using Navier-Stokes equations [9], mass balance in the form of continuity equations and energy conservation equations.

A particularly important problem in this description is the selection of an appropriate turbulence model. Because this flow

* SILESIAN UNIVERSITY OF TECHNOLOGY, FACULTY OF MATERIALS ENGINEERING AND METALLURGY, UL. KRASIŃSKIEGO 8, KATOWICE, POLAND,

\# Corresponding author: tmerder@polsl.pl

in certain regions of the tundish (the region of the liquid steel stream entering the tundish and outlets) is evidently turbulent, therefore the features of the steel and its moving state should take into account the phenomenon of turbulence.

This paper presents the results of modelling research of liquid flow through tundish. Numerical simulations were carried out using two numerical methods of turbulence description: RANS (Reynolds-Averaged Navier-Stokes) – model k-ε and LES (Large Eddy Simulation). The results obtained from CFD calculations were compared with the results obtained using the water model.

2. Research methodology

The trough, two-strand tundish of CC device without equipping the working area is under examination. This tundish has a bottom 0.15 m lower in the area of outlets. The steel is poured into the tundish through a ceramic shield placed in its plane of symmetry. The nominal capacity of the tundish is 60 tons. Fig. 1 shows the scheme of the tundish; whereas Table 1 presents the dimensions of the industrial tundish and the model made in 1:4 scale.
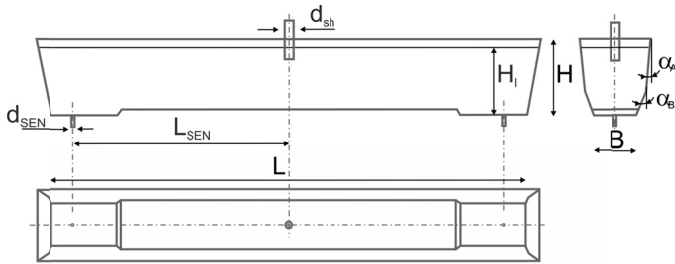

Fig. 1. Scheme of the tundish with designations of selected dimensions

TABLE 1

Dimensions of the 60 t tundish and the 1:4 scale water model

Parameter	Tundish	
	Scale (1:1)	Water model scale (1:4)
Volume – V, m^3	8.55	0.13
Tundish height – H, m	1.46	0.365
Filling level – H_l, m	1.2	0.3
Shroud diameter – d_{sh}, m	0.09	0.0225
Outlet diameter – d_{SEN}, m	0.052	0.013
Tundish bottom length – L, m	7.6	1.9
SEN position – L_{SEN}, m	3.8	0.95
Tundish bottom width – B, m	0.72	0.18
Angle of inclination of the walls – α_A, °	6	6
Angle of inclination of the walls – α_B, °	24	24

Considering the similar criterion, the research results obtained using water models can be transferred to a real object. If this can be possible, not only the geometrical similarity of the object and its model must be met, but also the dynamic similarity of the flow [10].

In the physical modeling of flow in tundish, the most commonly used criteria are the Froude number and Reynolds number. The Froude number is a criterion for flows caused by the force of gravity, e.g. flow with a free surface. The Reynolds number, on the other hand, defines the flow dynamics by comparing the ratio of the internal forces of the liquid to the frictional forces.

The overall similarity of flows, even in the cases outlined, is extremely difficult to achieve. An example may be the lack of simultaneous fulfillment of the criterion Fr and the criterion Re in models with a decreasing or enlarging linear scale S_L. The simultaneous fulfillment of both mentioned criteria is possible only in models with linear scale $S_L = 1$.

Therefore, partial similarity is considered sufficient by selecting one or two of the most important values in the analyzed flow, determining the so-called dominant criterion. In this research, the dominant criterion was the Froude number.

$$Fr = \frac{u^2}{g \cdot L} \qquad (1)$$

where:
u – velocity of the liquid,
g – acceleration of gravity,
L – characteristic length.

In the case of the dominant Froude criterion (1), using the scale method [11] to determine the similarity conditions, it is possible to determine the scale of velocity and time of the model liquid flow in the tundish model. Assuming, of course, that the same gravitational acceleration $g' = g$ works on the studied object, then in the model the condition of the similarity of Froude is:

$$Fr' = \frac{u'^2}{g' \cdot L'} = \frac{u^2}{g \cdot L} = Fr \qquad (2)$$

where ′ means the value concerning the model.

Time scale depends on the linear scale according to the equation:

$$S_t = \sqrt{S_L} \qquad (3)$$

where S_L – assumed linear scale,
whereas the velocity scale is determined from the following relationship:

$$S_V = \sqrt{S_L} \qquad (4)$$

From equation (2) it is obvious that for the scale of flow rate S_Q the following dependence is fulfilled:

$$S_Q = S_V \cdot S_L^2 = S_L^{5/2} \qquad (5)$$

Based on the criterion of similarity of Froude (1), on the basis of the relation (1-5) the velocity scale S_V, the flow time scale S_t and the flow rate scale S_Q for the model liquid were determined. These parameters were used to establish the kinetic similarity of the flow in the studied object (Table 2).

Numerical simulations were performed using the ANSYS Fluent program [12]. The values assumed for the calculation correspond to the conditions of the water model tests and are

listed in Table 2. The initial and boundary conditions (see Fig. 2) have been assigned to individual planes of the virtual model.

TABLE 2

The values accepted for model tests

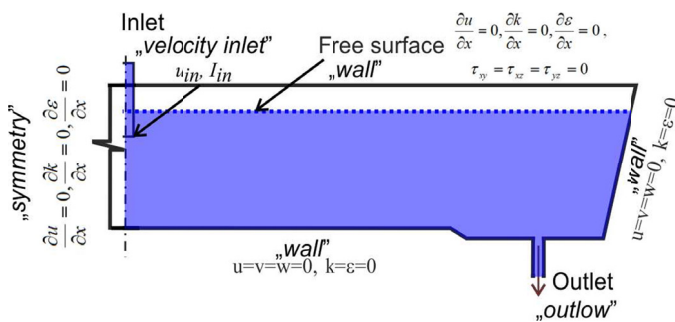| Parameter	Value
Volume flow rate – Q_V, m^3·s^{-1}	2.58×10^{-4}
Speed of the liquid at the inlet – u_{in}, m·s^{-1}	0.651
Water density – ρ, kg·m^{-3}	998.2
Kinetic viscosity of liquids – v, kg·m^2·s^{-1}	1×10^{-6}
Turbulent intensity inlet – I_{in}, %	5

Fig. 2. Initial and boundary conditions of the developed mathematical model

The working area of the object was mapped in the ANSYS SpaceClaim Direct Modeler preprocessor [13]. For the geometry of the tundish model, a computational grid densified in the region of the inlet and outlet was generated. The quality of calculated grids was checked using a criterial skew angle [12].

In order to find the optimal density of the calculation grid for the RANS method, the influence of the density of the calculation grid was analyzed, taking into account the speed of flowing water in the examined tundish. Based on this analysis, it was found that the mesh containing 228 thousand of cells is dense enough and can be used to perform numeric calculations.

In order to determine the effect of mesh density on the numerical solution using the LES method, the Pope's criterion was used [14]. When Pope's criterion Mp ≤ 0.2 [14], it is considered that the used computational grid (for the LES method) is appropriately densed. Areas that do not meet this dependence should be densed with additional nodal points. Based on this analysis, it was found that the mesh of 550,000 cells is dense enough and can be used to perform numeric calculations. A detailed description of the methodology was presented in paper [15].

CFD simulations describing the analyzed problem are a complex hydrodynamic issue in which, when reflecting reality, many features characterizing such flows should be taken into account. When constructing the appropriate mathematical model [16], a choice must be made between the available models and the initial and boundary conditions.

The flow of water in the model or liquid steel in tundish is characterized by a strong variation in turbulence in the whole working area of the reactor.

The direct method of solving turbulent Navier-Stokes equations is the DNS (Direct Numerical Simulation) method. However, it requires high computing power and a very long calculation time. This is due to the direct solution of all vortex structures (small and large). Alternatives to the DNS method are the Large Eddy Simulation (LES) method [17] and the RANS (Reynolds-Averaged Navier-Stokes) method [18].

The LES method separates vortex structures of large dimensions, which are solved directly (as in the case of DNS), from small sized vortex structures that are filtered, and their interaction with large eddy structures is modeled analytically. The SGS (Sub-Grid Scale) sub-mesh model [12] is used. This approach to solving the problem causes the use of a much more dense computing grid and a smaller time step than that used in the RANS method.

In the case of classic turbulence modeling (RANS method), for each size describing turbulent flow, a division into its average value and the fluctuation component takes place. This assumption leads to the presence in the N-S equations [16] of a member called the Reynolds stress tensor. It requires, in order to supplement the system of equations, modeling by using the turbulence model. In the majority of works in the field of CFD, concerning the analysis of liquid steel flow in tundish, the model [5,19-23] is commonly used, also models: k-ε RNG, k-ε Realizable [24-26] are applied.

In turbulent flows, which are analyzed flow, the presence of a wall has a significant impact on the flow field in its vicinity. The standard wall function (SWF - Standard Wall Functions) was used in calculations for the RANS method [16]. In calculations using the LES method, the modeling function (LESNWT – LES Near-Wall Treatment) was used [12].

In the calculations, the flow conditions were controlled depending on the density of the computational mesh in the boundary areas in such a way that the limit criteria were met.

During the numerical simulations on the outlet of tundish model, a change in marker concentration over time was recorded. By introducing the marker with the step method into the model, a residence time curve (RTD – Residence Time Distribution F-type) was obtained.

The partial differential equations described in the analyzed problem are nonlinear equations, and independent variables in them are coordinate functions of time and position. Therefore, the system of such equations is solved by numerical methods using sequences of appropriate procedures.

The initial stage of solving is the choice of the method of discretization of the system of equations, i.e. the replacement of the system of differential equations with an appropriate system of algebraic equations. In the ANSYS Fluent code, the control volume method is used for this purpose.

A criterion for convergence was set to be <10^{-6} on all variables and computations were carried out until the relative sum of residuals on all variables fell below the stipulated value. Velocity fields at steady state were first calculated and later they were employed to solve the mass transfer equation. For the species transport model, the solution was considered converged when

the residual for mass fraction was below 1×10^{-5}. The time step was 0.01 sec. Calculated 2000 sec. which allowed to generate the RTD curve.

Experimental research was carried out using the physical model of CC device. This model has a segmented structure. The individual structural elements belong to the main and auxiliary segments, which guarantee the fulfillment of the assumed functionalities of the test stand. The main segment of the model is this construction element in which phenomena relevant from the point of view of the expected results of the experiment occur. Similarity rules are met only in the main segment – the tundish model. The physical model view is shown in Fig. 3. This model is described in detail in [15].
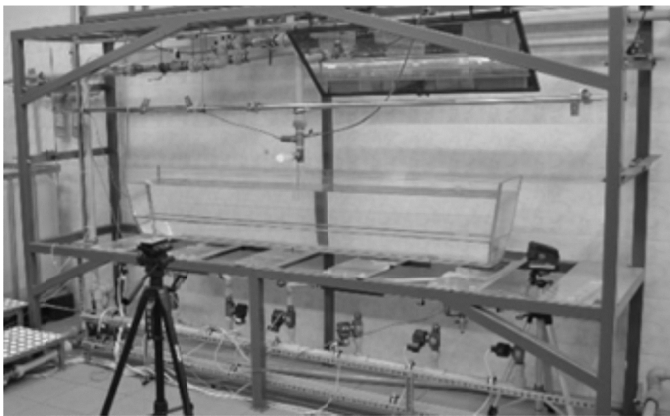

Fig. 3. View of physical model of steel continuous casting device

The method of measurement and the category of obtained results are of great importance in the implementation of model research. The experiments carried out on the water model were twofold: qualitative and quantitative. Material transparency (PMMA), from which the model object's walls are made, enables direct observation of the liquid flow, after introducing to it marker ($KMnO_4$) coloring water, which visualizes its movement. The movement of liquids, similarly to the mixing phenomena occurring in the tundish model, was examined qualitatively, recording the course of the experiment using high resolution video cameras. The image of flows was recorded in two planes – central and lateral (see Fig. 4). This arrangement of the cameras allowed for
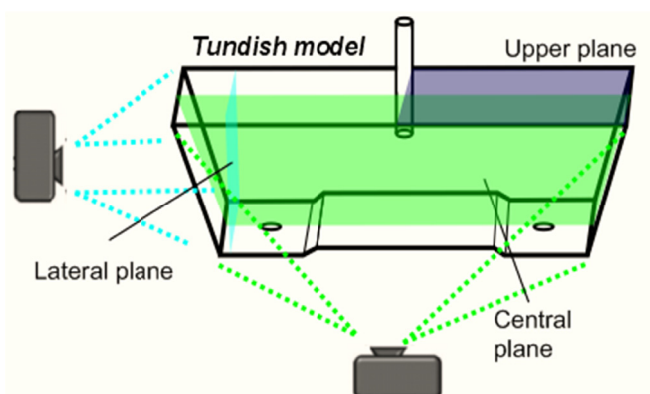

Fig. 4. Diagram of analyzed tundish model with marked control planes

uninterrupted observation of the model liquid circulation. The mirror placed over the tundish model allowed to additionally conduct observations from above.

In contrast, in quantitative experiments, determination of RTD characteristics (F-type curves), $KMnO_4$ was replaced with aqueous NaCl solution. The signals constituting the basis for drawing RTD curves are generated by conductometers that are installed in selected points of the working space of the tundish model (inlet and outlets). The voltage generated in half-second intervals by the conductometer is the equivalent of changes in the marker concentration in the water. This non-contact measurement does not affect the nature of the flow of the model liquid in the model's working space.

3. Results and discussion

The tests carried out in order to map the liquid flow structure in the tundish model were divided into two stages: qualitative analysis of the flow and mixing (process visualization) and quantitative analysis of the flow and mixing (by RTD curve F-type analysis).

For identical conditions of water flow in the tested object, available experimental material was confronted with data from CFD simulations.

Figs. 5 and 6 show selected CFD results (for RANS and LES methods) of liquid flow and experimentally determined distributions of marker concentration obtained experimentally (physical modeling).

Analysis of the qualitative compatibility of CFD simulations with the results of the experiment reveals that the RANS method is better suited to identifying the character of the model liquid flow in the working space of tundish model.

Fig. 7 shows a summary of RTD characteristics (F type) obtained from CFD (for RANS and LES methods) and experimental measurements. They were used for quantitative assessment.

In order to directly compare the characteristics of the physical modeling, the values of the marker concentration were converted to a dimensionless value, so its minimum value is 0, and the maximum value equal to 1. The dependence described in detail in [5,15] was exploited for this purpose.

Analyzing the curves shown in Fig. 7, noticeable differences are visible for the LES method. In the initial phase, i.e. from the moment $t = 0$ to the time $t = 400$ seconds, the curve deviates significantly from the others. There is a slight difference between measured and calculated data for individual curves. These differences are revealed in the form of a shift curves, but the rate of marker concentration increase is largely consistent. The observed shift of curves obtained for the water model as a function of time in relation to the numerical results (time of reaching the marker to outlets – see Table 4) is caused by: a difficult to grasp the moment of start in water modeling and numerical idealization of marker introduction.

The characteristics obtained on the experimental and CFD-RANS routes show high quality compliance.
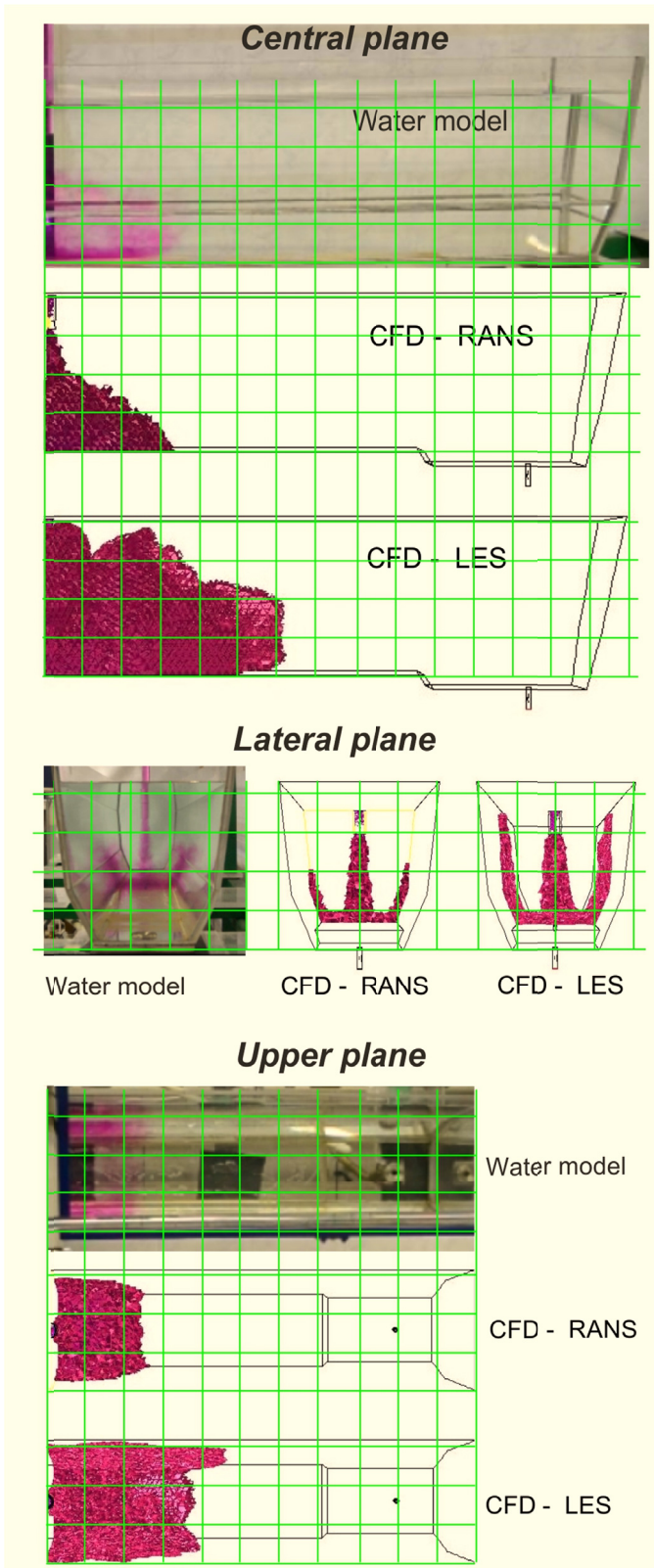
Fig. 5. Comparison of tracer distribution in model tundish for time 5 seconds

Fig. 6. Comparison of tracer distribution in model tundish for time 30 seconds

For comparison of the labeled marker concentrations obtained from physical modeling with the concentrations obtained from numerical modeling, a relative error was calculated for each determination, defined as follows:
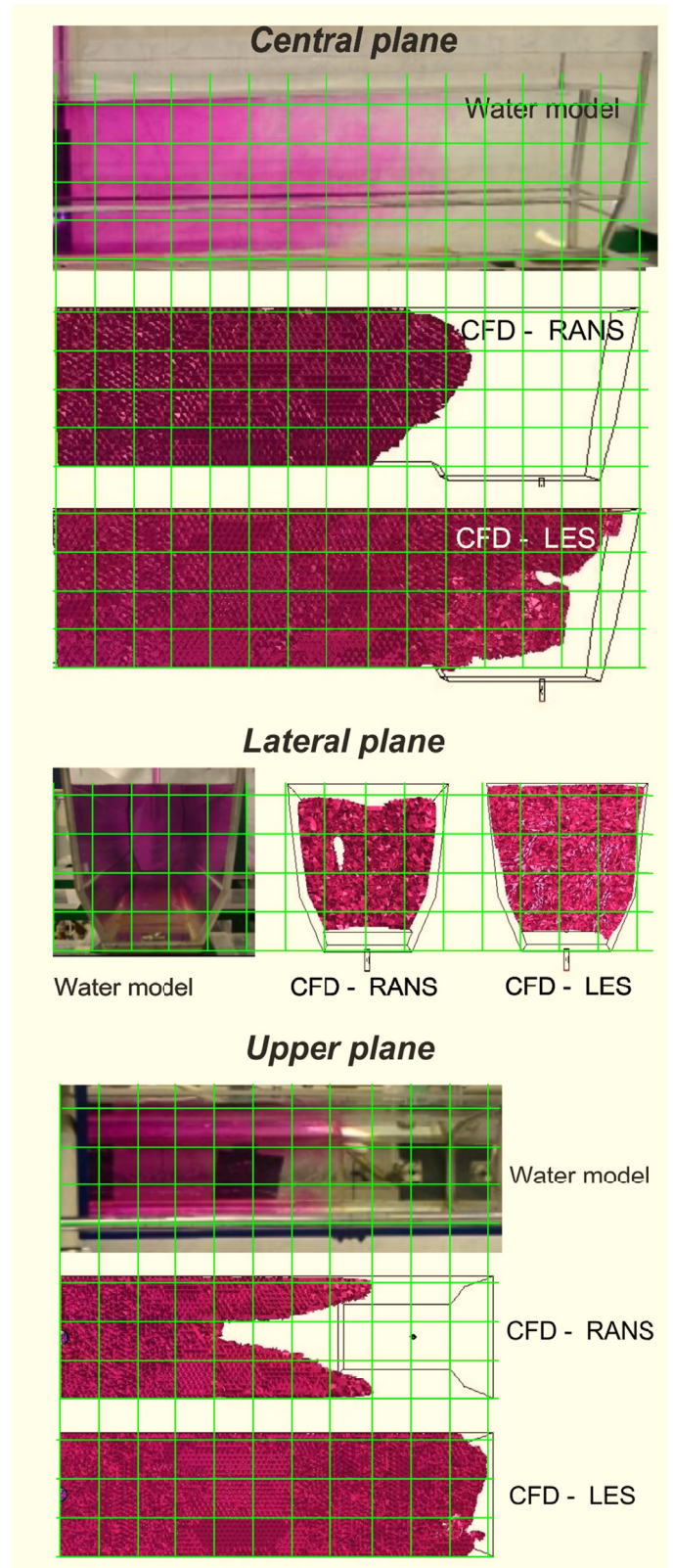
$$\delta_i = \left(C_i^{\exp} - C_i^{sym} \middle/ C_i^{\exp} \right) 100\% \qquad (6)$$

where:

$C^{\exp}$ – tracer concentration of the laboratory experiment,
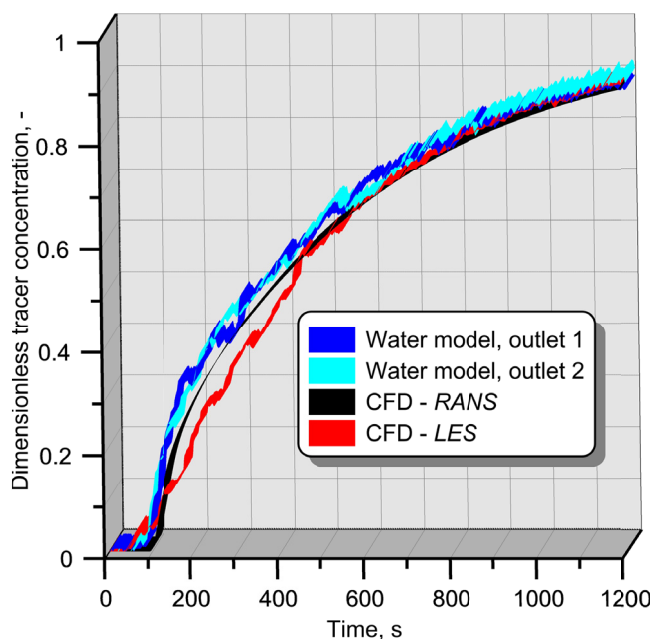C^{sym} – tracer concentration of CFD modeling.

Fig. 7. Summary of RTD F-type characteristics obtained in the water model and CFD simulations (RANS and LES)

TABLE 4

Time to reach the marker for outlets

Method	Time, s
CFD-RANS	92
CFD-LES	65
Water model-outlet 1	85
Water model-outlet 2	83

The error value (see Table 5) for CFD-LES calculations is much higher than for CFD-RANS. The error value for the RANS method does not exceed 11%. It can be concluded that the mathematical model of the liquid flow based on the RANS method well reflects changes in the marker concentration (flow) in the studied tundish model. This statement justifies the fact that an 11% error is generally considered acceptable when assessing reliable flow characteristics.

TABLE 5

Relative error of tracer concentration (CFD and laboratory experiment)

		Numbering of outlets			
		1	2	for two outlets	
				1	2
Error value, %	CFD-RANS				
	Minimum	−1.0	1.7	−1.0	11.0
	Maximum	10.5	11.0		
	CFD-LES				
	Minimum	−2.1	−0.5	−2.1	30.0
	Maximum	30.0	28.8		

Analyzing the presented comparison of results in terms of the qualitative and quantitative compatibility of CFD simulations

with the results of a laboratory experiment, it should be stated that in the description of the turbulent nature of the flow of the model liquid in the tundish model, the RANS method is better suited.

4. Summary and statements

CFD simulations of liquid steel flow in a tundish (or water in the model) is a complex hydrodynamic issue in which the reality should be reflected in a number of features characterizing such flows. Constituting the appropriate mathematical model, you have to choose among the available hydrodynamic models of phenomena and possible initial and boundary conditions. Usually, such a choice is a compromise between the precision of the solution and the time of calculation. It must therefore be preceded by testing calculations that allow estimating the level of errors caused by the assumed simplifications. It is also good practice to use a different test method (physical modeling or partial measurements on a real object) with numerical calculations in order to supplement and verify the obtained results.

In the course of a comparative analysis of the empirical results of measurements from the water model with the results of numerical simulations – identifying the three-dimensional turbulent nature of liquid flow in the working space of the examined tundish – it was shown that the obtained results are significantly influenced by the appropriate choice of the turbulence model. A qualitative and quantitative comparison of the test results obtained from both research techniques showed that the results of experimental measurements were closer to the results of CFD simulations using the RANS method.

Based on the model tests carried out, it was found that:
−   CFD simulations using the k-e turbulence model, allow to identify the nature of liquid flow, as well as to determine the volume of flow zones in the tundish under test with a possible, acceptable error loading such designation, not exceeding 11%,
−   for the RANS method, the calculation time is twice shorter to achieve a convergent solution.

REFERENCES

[1]   A. Braun, M. Warzecha, H. Pfeifer, Metall. Mater. Trans. B **41**, 3, 549-559 (2010).

[2]   K. Michalek, M. Tkadleckova, K. Gryc, P. Klus, Z. Hudzieczek, V. Sikora, P. Strasak, International Conference on Metallurgy and Materials – METAL 2011, 143-149 (2011).

[3] M. Tkadleckova, K. Gryc, P. Machovcak, K. Michalek, L. Socha, M. Kovac, Materiali in Technologije **46**, 4, 399-402 (2012).

[4] M. Saternus, J. Botor, Metalurgija **48**, 175-179 (2009).

[5] M. Warzecha, T. Merder, P. Warzecha, G. Stradomski, ISIJ Int. **53**, 11, 1983-1992 (2013).

[6] A. Cwudziński, Archives of Metallurgy and Materials **60**, 3, 1581-1586 (2015).

[7] G.Wang, M.Yun, Ch. Zhang, G. Xiao, ISIJ Int. **55**, 5, 984-992 (2015).

[8] F. He, L. Zhang, Y. Xu, China Foundry **13**, 3, 166-175 (2016).

[9] J.M. Kay, R.M. Nedderman, Fluid mechanics and transfer processes. Cambridge University Press, Great Britain 1985.

[10] K. Michalek, Využití fyzikálního a numerického modelování pro optimalizaci metalurgických procesů, VŠB Ostrava, 2001, ISBN 80-7078-861-5 (in Czech).

[11] L. Müller, Zastosowanie analizy wymiarowej w badaniach modeli, PWN, Warszawa 1983 (in Polish).

[12] ANSYS Fluent ver. 16.0 – user's guide, Canonsburg, USA 2015.

[13] ANSYS SpaceClaim Direct Modeler – user's guide, Canonsburg, USA 2013.

[14] S.B. Pope, New Journal of Physics **6**, 1, 1-24 (2004).

[15] T. Merder, Modelowanie numeryczne i fizyczne rozkładu wtrąceń niemetalicznych podczas przepływu ciekłej stali przez kadź pośrednią urządzenia COS, Gliwice 2018.

[16] J.F. Wendt, Computational fluid dynamics. Springer-Verlag, Berlin 1996.

[17] B.E. Launder, D.B. Spalding, Computer Methods in Applied Mechanics and Engineering **3**, 269-289 (1974).

[18] P. Sagaut, Large-Eddy Simulation for incompressible flows: An introduction. Springer-Verlag, Berlin/NY 2001.

[19] D.C. Wilcox, Turbulence modeling for CFD. DCW Industries, Inc., La Canada, California 1993.

[20] A. Kumar, D. Mazumdar, S.C. Koria, ISIJ Int. **48**, 1, 38-47 (2008).

[21] S.K. Mishra, P.K. Jha, S.C. Sharma, S.K. Ajamani, Int. Journal of Minerals, Metallurgy and Materials **18**, 5, 535-542 (2011).

[22] T. Merder, M. Saternus, M. Warzecha, P. Warzecha, Metalurgija **53**, 3, 323-326 (2014).

[23] P. Warzecha, M. Warzecha, T. Merder, Metalurgija **54**, 3, 462-464 (2015).

[24] R. Schwarze, F. Obermeier, J. Hantusch, A. Franke, D. Janke, Steel Research Int. **72**, 5-6, 215-220 (2001).

[25] H.J. Odenthal, R. Bolling, H. Pfeifer, Steel Research Int. **74**, 1, 44-55 (2003).

[26] Q. Hou, Z. Zou, ISIJ Int. **45**, 3, 325-330 (2005).