T. MERDER, M. SATERNUS, M. WARZECHA, P. WARZECHA

ISSN 0543-5846
METABK 53(3) 323-326 (2014)
UDC – UDK 669.186:518.1:531.3=111

VALIDATION OF NUMERICAL MODEL OF A LIQUID FLOW IN A TUNDISH BY LABORATORY MEASUREMENTS

Received – Prispjelo: 2013-10-10
Accepted – Prihvaćeno: 2014-02-20
Original Scientific Paper – Izvorni znanstveni rad

The article presents results of physical and numerical modelling of steel flow through a tundish of continuous casting machine. In numerical calculations the influence of mesh density was tested and the correctness of the flow description in the near-wall region was checked using Standard Wall Function model. Obtained results were verified using experimental results of velocity field (PIV method) coming from a water tundish model.

Key words: tundish, numerical modelling, physical modelling, PIV method

INTRODUCTION

If the research was to be carried out on metallurgical object, it would be quite difficult or sometimes even impossible due to high temperatures of the process and the fact that liquid (steel) is non-transparent. That is why, such research is conducted using physical (water) models which are usually scaled-down. Today numerical modelling is commonly applied. In numerical modelling of different kinds of flows CFD (Computational Fluid Dynamics) programs are used; they are based on the basis of numerical mechanics of liquids. CFD method is one of the elements of liquid mechanics which is grounded on both numerical methods as well as algorithms used for the analysis and solution of different problems describing flow of liquids and gases. Applying CFD technique virtual model of an apparatus or a process is built using laws of liquid movement in the form of mathematical equations describing the physics of the flow. CFD technique considerably shortens the time from project to production. To achieve this aim faster calculating machines are used; the capacity of processor shortens the time of research. Despite of applying strong calculating processors and continuously improving mathematical models, only approximate view of the studied problem is obtained. Therefore results of experimental research carried out on water models apart from experimental industrial values are commonly used as the base for verification of numerical simulations [1-6].

Presented results deal with analysis of steel flow through a tundish of continuous casting (CC) machine; this analysis was made using both types of modelling. In numerical calculations (CFD) the influence of mesh density was tested and the correctness of flow description in the near-wall region – in this case Standard Wall Function (SWF) model was used. Obtained results were verified using experimental results of a velocity field (PIV method) coming from water tundish model.

T. Merder, M. Saternus: Silesian University of Technology, Department of Metallurgy, Katowice, Poland
M. Warzecha: Czestochowa University of Technology, Department of Metals Extraction and Recirculation, Czestochowa, Poland
P. Warzecha: Czestochowa, Poland

OBJECT AND METHODOLOGY OF RESEARCH

One-strand tundish with 16 Mg capacity was under study (physical model at 1:3 scale).This tundish features flat bottom without any equipment of the working zone. In industrial conditions such tundish casts 137 Mg/h of steel. Figure 1 presents the scheme of the tundish model and Table 1 shows its dimensions.

In the physical modelling the specialistic and new apparatus based on Dantec Flow-Map-DPIV-1100 system was used [7]. Such system uses laser-optic technique to measure the velocity field of a liquid. The view is registered by means of digital camera, and then it is

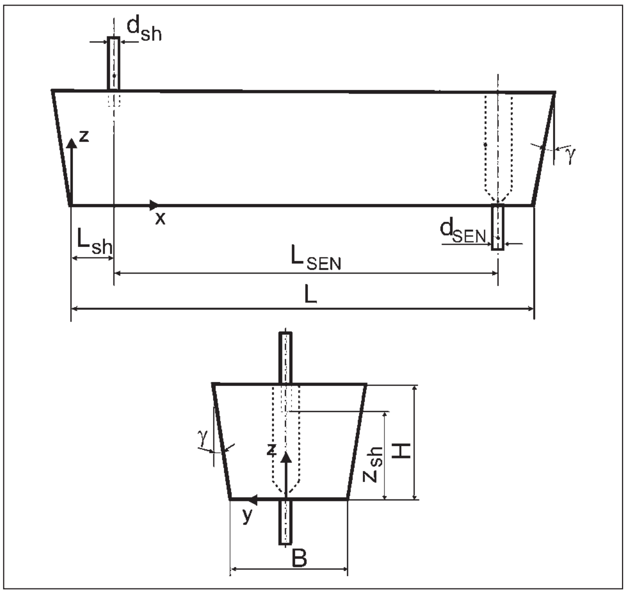

Figure 1 Schematic illustration of the shape and size of the water model tundish

Table 1 **Dimensions of the water tundish model (1:3 scale)**

Parameter	Water model at the 1:3 scale
Volume of tundish at filling level H / m^3	0,0084
Tundish length L / m	1,047
Tundish width B / m	0,26
Filling level H / m	0,266
Inclination of the side walls α / °	7
SEN position L_{SEN} / m	0,962
Shroud position L_{SH} / m	0,122

converted using special software. It enables to obtain precise distributions of velocity vectors at the planes optionally intersecting the examined object.

For numerical modelling of a liquid flow through a tundish model, the commercial calculating code ANSYS Fluent was used. It is based on Navier-Stokes` differential equations. For modelling the turbulence, k-ε model was applied because the considered flow is turbulent [8]. This model is widely used in problems of engineering [1-6,9,10]. For the geometry of the tundish model a block-structural mesh was generated, condensed in the area of inlet and outlet. The side walls of the model and also the bottom of the tundish were modelled as a stationary walls using Standard Wall Functions (SWF) [6].

Water parameters for calculations were: density (998,2 kg·m^{-3}) and viscosity (0,001 kg·m^{-1}·s^{-1}).

Non-linear system of partial differential equations was discretized using the finite volume technique in a computational domain and solved with the help of boundary conditions using ANSYSFluent code. All numerical simulations were carried out with the use of double-precision solver (3ddp) and the second-order spatial discretization scheme. For describing the pressure-velocity coupling, the SIMPLEC algorithm was selected. The mathematical simulations were run on an INTEL CORE i7 computer processor.

RESULTS AND DISCUSSION

Numerical modelling

Numerical calculations require special care during preparing mesh (i.e. flow of liquid steel in a tundish). Discretization of the calculating zone necessary for this purpose and for examined tundish is done by means of mesh generating program.

Taking into consideration the convergence and precision of calculations the best choice would be applying homogenous structural mesh in the whole calculating domain. However the geometrical complexity of an object working area inclines to apply hybrid meshes coming from coupling the structural meshes with non-structural ones.

One of the criteria used for checking the correctness of calculating mesh is criteria of the skewness angle defined by the following relationship [11]:

$$Q_{EAS} = \max\left\{\frac{\beta_{max}-\beta_{eq}}{180-\beta_{eq}}, \frac{\beta_{eq}-\beta_{min}}{\beta_{eq}}\right\} \qquad (1)$$

where: β_{max} and β_{min} are the maximal and minimal angles (in grades) between edges of element, β_{eq} is the angle corresponding to the ideal cell, its value for cubic cells is 90°.

Value of Q_{EAS} coefficient normalized in the range of $\langle 0;1 \rangle$ should be lower than 0,75 [11], which determines the acceptable skewness angle of generated mesh.

Density of calculating mesh essentially influences the research results. In places where high gradients of modelling liquid velocity is observed, well prepared mesh should be locally condensed. Calculating mesh in the studied object was condensed near the tundish inlet and outlets that means in the area of increased level of turbulence and velocity.

To search optimal density of calculating mesh the meshes containing 100, 180, 290 and 320 thousand control volumes were assumed. The influence of mesh density was analyzed considering the velocity of flowing medium along the tundish length and width. Figure 2 presents the example of obtained results.

It should also be mentioned that the more elements the longer calculating time needed for obtaining the convergent solutions. Therefore, it is necessary to find optimal numbers of calculating mesh grids.

It comes from Figure 2 that mesh containing not less than 290 thousand cells is dense enough and can be used for numerical calculations.

The next important stage of research is the description of turbulent flow (the flow that exists in tundish). Determination of liquid movement parameters in the next-wall region enforces applying very dense calculating mesh to reveal characteristic distributions in so called viscous sublayer. This involves considerable elongation of calculation; that is why in many technical applications so called „wall function", which utilizes analytical solution in this area is used.

The model of flow, used in this work, describes turbulence in the tundish basing on Launder and Spalding`s

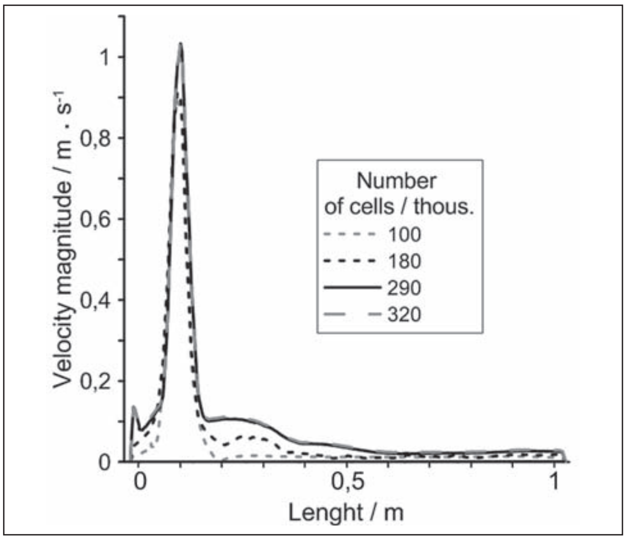

Figure 2 Velocity of water in tundish model in the function of calculating mesh density in the intersection crossing the centre of the object and at the height 0,5 H

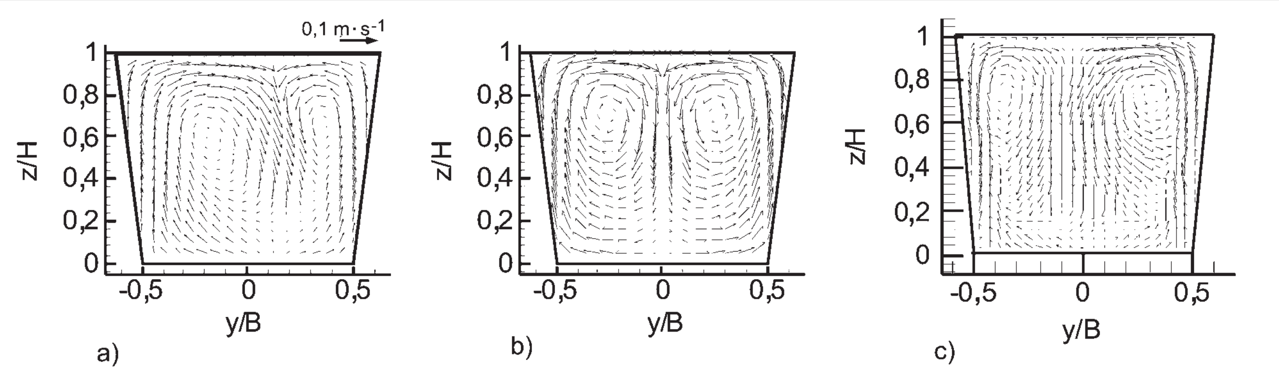

Figure 3 Fluid flow field (plane x/L=0,5): a) CFD $y^+ = 220$, b) CFD $y^+ = 58$, c) measured with DPIV [7]

postulate [8], in which standard wall function (SWF) is applied. According to it the profile of velocity in turbulent near-wall region is described by logarithmic dependences [12]. Therefore to calculate the velocities in near-wall region standard wall function is applied and the following expression is used:

$$U^+ = \frac{1}{\kappa}\ln\left(Ey^+\right) \qquad (2)$$

where: E – empirical constant (= 9,793),
κ – von Karman`s constant (= 0,4187),
y^+ – normalized distance of the nearest grid from the rigid wall.

Applying standard wall function requires, however, fixing the distance of first mesh grid from the wall in the zone of logarithmic distribution. According to the instruction of ANSYSFluent program such distance can be controlled and also fixed on the base of y^+ parameter evaluation. This parameter should be in the range from 30 to 60 for the SWF model. Such parameter can be calculated from the following relationship:

$$y^+ \equiv \frac{\rho\, u_t y}{\mu} \qquad (3)$$

where: ρ – density,
$u_t = \sqrt{\tau/\rho}$ – velocity of near-wall flow friction,
μ – viscosity,
τ – shearing stresses on the wall,
y – distance of the nearest grid from the rigid wall.

Model SWF was used in numerical simulations. During calculations the conditions of steel flow were controlled depending on mesh density in near-wall areas in such a way that border criteria could be fulfilled. Such control was done basing on value of y^+ parameter that was determined for a given density of calculating mesh.

Figure 3 presents results of numerical simulations that identify the flow in the examined object obtained for different values of y^+ parameter. In Figure 3a the dimensionless parameter (y^+) is away from the borders of applicability of the logarithmic wall function ($y^+ = 220$), whereas in Figure 3b the criterion of correctness is fulfilled ($y^+ = 58$).

For comparison the results of experimental research coming from physical model (for identical plane) were also presented in Figures 3c. Figures 3a and 3b present considerably different view of liquid movement when the value of y^+ parameter exceeds the allowable limits. That is why when applying turbulence model k-ε from SWF the values of y^+ parameter in conducted research should be accurately controlled

Physical modelling

Experimental measurements of a flow field enable to compare them directly with the results obtained numerically. Figures 4 and 5 present the distribution of velocity vectors (fluid flow field) on the planes crossing tundish on the axis inlet / outlet (y/B = 0), and near the side wall (y/B = 0,5).

Water flowing into the tundish model through the lining pipe is rebounded and then directed in the direction of side walls. On the left side circulative movement is created (Figure 4) where water flux rebounded from the bottom reaches the height z/H = 0,6. The flux of water flowing in the direction of outlet hits the returnable flux creating another area of liquid circulation in the tundish model bottom (x/L = 0,4 ÷ 0,6). The centre of such vortex is placed on the right from the middle of a distance between inlet and outlet of the model. Very similar mapping of flow field was obtained from CFD calculations.

When analyzing the flow at the intersection along the side wall of the tundish model (see Figure 5), it was ob-

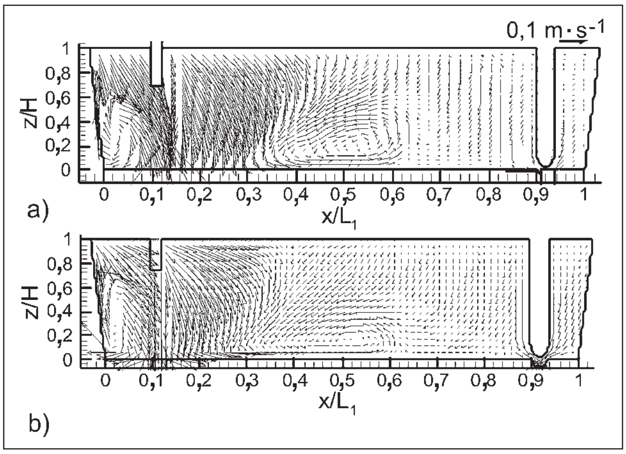

Figure 4 Fluid flow field (plane y/B = 0): a) measured with DPIV [7], b) CFD

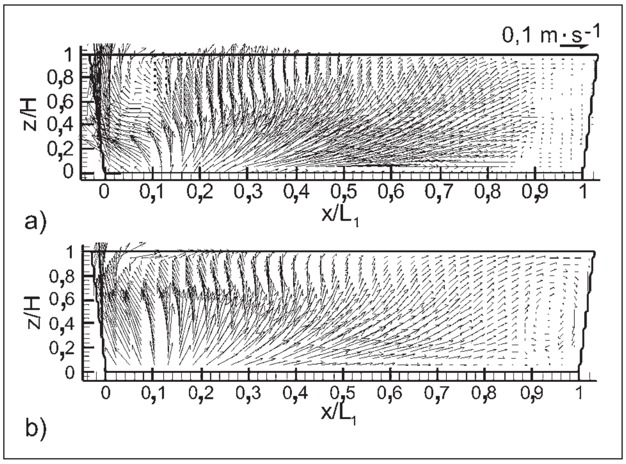

Figure 5 Fluid flow field (plane y/B = 0,5): a) measured with DPIV [7], b) CFD

served that liquids flow around walls directing to the upper part and outlet. Whereas, the direct flow of liquid from the area of inlet to the area of outlet is seen near the tundish bottom. Very similar mapping of flow field was obtained from CFD calculations.

To sum up, the obtained flow fields (see Figures 3 and 5) show the compatibility of CFD calculations with experimental measurements.

CONCLUSIONS

Physical and numerical models enable to map correctly the movement of a medium flowing through the tundish.

The numerical solution requires high carefulness in preparing input file. The most important elements to be considered are: correct mapping of the calculating area geometry; calculating mesh; kind of model; physical parameters of a liquid; correct determination of the kinds of initial boundary conditions. That is why to verify the correctness of liquid movement description, obtained from CFD calculations should be confronted with experimental results.

Presented results pointed that quality of calculating mesh influences the correctness of obtained results. It should be remembered that when applying the turbulence model (k-ε) with standard wall function, it is necessary to control the value of y^+ parameter. When the value of such parameter exceeds the allowable limits, the calculated view of the liquid movement can be deformed and may not map the real conditions of the process.

REFERENCES

[1] F. Shen, J. M. Khodadadi, S. Pine, X. Lan, Metal. and Mater. Trans. B, 25B (1994) 3, 669-680.
[2] A. Robert, D. Mazumdar, Steel Research Int., 71 (2001) 3, 97-105.
[3] S. Lopez-Ramirez, J. J. Barreto, P. Vite-Martinez, J. A. Romero-Serano, D. C. Valencia, Metal. and Mater. Trans. B, 35B (2004) 5, 957-966.
[4] K. Michalek, K. Gryc, J. Morávka, Metalurgija, 48 (2009) 4, 215-218.
[5] T. Merder, J. Pieprzyca, M. Warzecha, Metalurgija, 48 (2009) 3, 143-146.
[6] M. Warzecha, T. Merder, H. Pfeifer, J. Pieprzyca, Steel Research Int., 81 (2010) 11, 987-993.
[7] M. Warzecha, Hutnik - Wiadomości hutnicze, 10 (2010) 77, 571-574.
[8] B. E. Launder, D.B. Spalding, Methods in Applied Mechanics and Engineering, 3 (1974), 269-289.
[9] R. Przyłucki, A. Smalcerz, Metalurgija 52 (2013) 2, 235-238.
[10] M. Saternus, T. Merder, P. Warzecha, Solid State Phenomena, 176 (2011), 1-10.
[11] Manual ANSYS Fluent, (2012).
[12] D. C Wilcox, Turbulence modeling for CFD, La Canada, CA: DCW Industries (1993).

Note: The responsible translator for English language is M. Kingsford, Katowice, Poland