www.czasopisma.pan.pl PAN POLSKA AKADEMIA NAUK www.journals.pan.pl

A R C H I V E S O F M E T A L L U R G Y A N D M A T E R I A L S

Volume 59 2014 Issue 2

DOI: 10.2478/amm-2014-0134

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

POSSIBILITIES OF 3D MODEL APPLICATION IN THE PROCESS OF ALUMINIUM REFINING IN THE UNIT WITH ROTARY IMPELLER

MOŻLIWOŚĆ ZASTOSOWANIA MODELOWANIA 3D DO PROCESU RAFINACJI ALUMINIUM W REAKTORZE Z ROTOREM

Both primary and secondary aluminium have to be refined, especially by barbotage process. To know better the mechanism of blowing argon through aluminium in reactors with rotary impellers, numerical modelling is applied. It allows to obtain useful information like: the level of velocity field or participation of gaseous phase. However, numerical analysis requires choosing the proper model which would describe the physical phenomena occurring in the process. So, AnsysFluent code was used in the research. It allows to calculate the two-phase liquid flow for the 2D and 3D co-ordinate systems. Results of calculations for the 3D case can describe more accurately the spatial picture of the movement trajectory of the blown gas bubbles. Calculations were done for the flow rate of refining gas equal 5 dm^3/min and for two cases: when there was no rotation and with 300 rpm rotary impeller speed.

Keywords: aluminium, refining, numerical modelling

Zarówno aluminium pierwotne, jak i wtórne musi zostać poddane rafinacji, np. poprzez barbotaż. W celu poznania mechanizmu przedmuchiwania aluminium argonem w reaktorach z rotorami stosuje się modelowanie numeryczne. Pozwala ono uzyskać takie informacje jak: określenie pól prędkości czy udziału fazy gazowej. Modelowanie matematyczne wymaga jednakże odpowiedniego modelu określającego warunki fizyczne panujące w procesie. Tak więc, do obliczeń zastosowano kod AnsysFluent, który pozwala na obliczenia przepływów dwufazowych dla dwu- (2D) i trójwymiarowego układu współrzędnych (3D). Wyniki obliczeń dla układu 3D mogą bardziej dokładnie opisać trajektorie ruchu wdmuchiwanych pęcherzyków gazowych. Obliczenia przeprowadzono dla natężenia gazu 5 dm^3/min przy braku rotacji oraz przy prędkości obrotowej 300 obr/min.

1. Introduction

Today, both primary and secondary aluminium contains many impurities such as hydrogen or nonmetallic and metallic inclusions. To remove from liquid metal mainly hydrogen which causes porosity and also parts of metallic and nonmetallic inclusions, aluminium is blown by inert gas bubbles e.g. argon. This process is characterized by high dynamics of run because of the quick mass transfer between phases. If the refining process via blowing with argon has to be efficient, it is necessary to obtain appropriate size reduction of gas bubbles and their dispersion in liquid metal [1-3]. Thus, hydrodynamic conditions of the process are very important. They can be determined by the physical modelling. However, the flow of mass and gas is not fully shown by this modelling. The complementation and extension of such type of modelling is the numerical simulation [4].

Today numerical modelling is commonly applied to solve engineering problems in different fields of industry [5-9]. It is possible due to regular development of computational methods of modelling together with the increasing computer power. Thus, a very comfortable and precise enough research tool is obtained. Results of calculations allow quicker and better to verify the design ideas, to optimize the design process and to save time and money.

2. Description of URO-200 reactor

The aluminium refining process by blowing the inert gas through aluminium can be conducted in continuous and bath reactors. The inert gas can be introduced to liquid metal in different ways: lances, porous plugs, special nozzles and rotary impellers. The last solution becomes more and more popular. In Poland one of the most popular reactors of such types is the URO-200 reactor designed by IMN-OML in Skawina. The problem connected with this type of rectors is to obtain the uniform dispersion of gas bubbles in the whole volume of liquid. The following processing parameters influence it: flow rate of refining gas and mainly the rotary impeller speed. The choice of these parameters ensures the optimization of aluminium refining process. Fig. 1 presents the geometry and dimensions of the URO-200 reactor.

* SILESIAN UNIVERSITY OF TECHNOLOGY, 8 KRASIŃSKIEGO STR., 40-019 KATOWICE, POLAND

** CZESTOCHOWA UNIVERSITY OF TECHNOLOGY, 19 ARMII KRAJOWEJ STR., 42-200 CZĘSTOCHOWA, POLAND

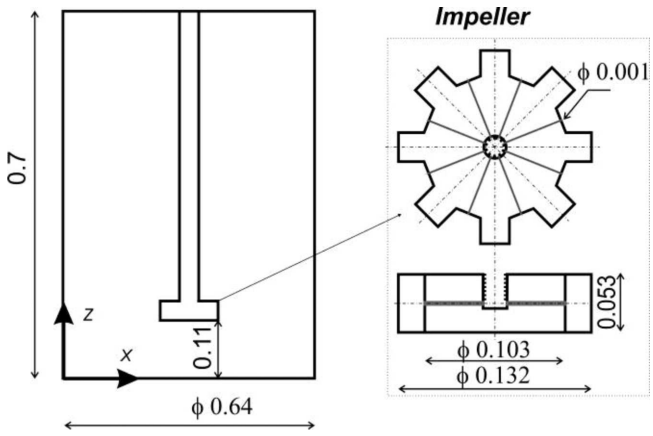

Fig. 1. The shape and size of URO-200 reactor

3. Numerical modelling

Numerical solution of analyzed problem is followed by applying the multiphase (diphase) model of flow. Many ways of mathematical descriptions of multiphase flow are known, for example one method of such problem solution is to describe one of the phases in Lagrange way. Movement equations are calculated directly for every particle, therefore this method demands relatively longer calculation time. More effective, taking into account the calculation time, is the description of diphase flow using Euler's method. Then for every phase conservation equation of mass and momentum are solved taking into consideration the appropriate term, the main aim of which is to conjugate both phases. However, this method is followed by the necessity of applying the simplified assumptions and also formulating appropriately the boundary conditions. That is why this method is quite commonly applied in solving the diphase flow [10-12]. Numerical analysis requires choosing the proper model describing the physical phenomena occurring in the process. Such choice is always the compromise between the accuracy of solution and demanded calculation time. The complexity of mathematical model and the demand of calculation correctness require to use the effective tool for numerical solution, so the commercial code AnsysFluent was used.

Mathematical model describing the flow of liquid in the examined object consists of the system of differential equations: the continuity of flow equation, momentum conservation equation and equation describing the turbulence structure of the liquid movement [13]. For modelling the turbulence model k-ε (creating by Launder and Spalding [14]) was used; it is commonly applied in solutions of engineering problems [15-17]. Model VOF (Volume of Fluid) was used to model the multiphase (diphase) flow [18]. This model is based on the Euler's description and is designed for two or more immiscible flows. It is also possible to determine the flows with free surface. Phases in this model are treated as continuous phases that do not penetrate mutually. VOF model needs the solution of movement equations for each phase (liquid and gaseous). Additionally, it takes into account forces of surface tension and adhesion by the source elements in momentum equations.

The volume of cell which contains fluid can be defined by the following function:

$$\varepsilon_{k(cell)} = \iiint\limits_{cell} \varepsilon_k(x,y,z)dxdydz \Big/ \iiint\limits_{cell} dxdydz \qquad (1)$$

The volume fraction determined in this case can take one of three forms: when $\varepsilon_k = 0$, so cell does not contain k fluid, when $\varepsilon_k = 1$, then cell is full of k fluid, and when $0 < \varepsilon_k < 1$, that means the cell contains the interface between the fluids.

Dislocation of the interfacial can be determined by solving continuity equation in the following form:

$$\frac{\partial \varepsilon_k}{\partial t} + u_j \frac{\partial \varepsilon_k}{\partial x_i} = 0 \qquad (2)$$

In numerical simulations three cases were studied (Table 1). In the first case, called Case A, the analyzed problem was simplified to 2D case – it helped to check the correctness of assumed simplifications and boundary conditions. Cases second (Case B) and third (Case C) concerned 3D geometry. These cases differed from the applied calculation mesh. For Case B it was a stationary calculation mesh with the application of periodical boundary conditions, whereas for Case C the movable calculations mesh was applied, which should additionally simulate the rotary impeller movement.

In analyzed cases the symmetrical flow distribution towards planes passing through the centre of the object was assumed, therefore the calculation domain consisted of the sector of a flow field. Additionally, calculation mesh was condensed in the area of impeller, that is the area of blowing gas into the metal. Problems were solved numerically by the methods of volume control. Fig. 2 shows the geometry of studied cases with applied boundary conditions. Table 1 also presents parameters and setting of the numerical calculation for studied cases.

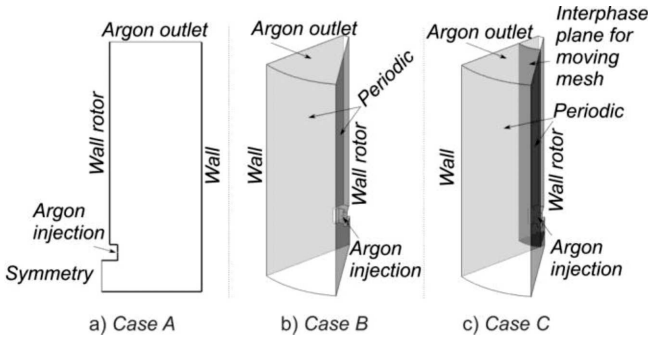

Fig. 2. Geometry and boundary conditions applied in simulations for analyzed test cases

TABLE 1

Parameters and solver settings for analyzed test cases

Case	Case A	Case B	Case C
Geometry	2D	3D	3D
Computational domain	1/2	1/8	1/8
Mesh typ	hexa	hexa/tgrid	hexa/tgrid
Number of cells	13180	69475	218453
CFD code	AnsysFluent 13	AnsysFluent 13	AnsysFluent 13
Time dependency	unsteady	unsteady	unsteady
Time step size [s]	0.001	0.0002	0.0001
Multiphase model	Volume of Fluid	Volume of Fluid	Volume of Fluid
Phase type	1 phase – water 2 phase – argon	1 phase – water 2 phase – argon	1 phase – water 2 phase – argon
Turbulence	RANS: k-epsilon	RANS: k-epsilon	RANS: k-epsilon
Near-wall treatment	Standard Wall Function	Standard Wall Function	Standard Wall Function
Pressure velocity coupling	PISO	SIMPLE	SIMPLE
Pressure discretisation	PRESTO!	PRESTO!	PRESTO!
Momentum discretisation	second order upwind	second order upwind	second order upwind
Conditions of modelling media			
Water density [kg·m^{-3}]	998.2	998.2	998.2
Water viscosity [kg·m^{-1}·s^{-1}]	0.001	0.001	0.001
Argon density [kg·m^{-3}]	1.6228	1.6228	1.6228
Argon viscosity [kg·m^{-1}·s^{-1}]	2.125e-05	2.125e-05	2.125e-05
Conditions of refining gas inlet			
Flow rate of refining gas, [dm^{3}min^{-1}]	5	5	5
Conditions of rotary impeller working			
Rotary impeller speed [rpm]	–	300	300

4. Results of the research

As a result of calculation, the forecasted argon volume fraction (Fig. 3) and velocity magnitude distribution (Fig. 4) were obtained for the studied cases. Additionally, in Fig. 4 with forecasted velocity distribution, the flow lines were shown in order to visualize liquid (water) movement better. Results were presented for different calculation times: after 2, 4, 6 and 8 seconds.

5. Summary

To get to know mechanism of the aluminium refining process it is necessary to optimize its course. Conducting the research in industrial conditions is rather hard as it disturbes the normal run of the production. Therefore, the best way to get such knowledge is to model analyzed problem using different methods. Optimal solution is to join physical and numerical modelling. Such modelling complements and verifies each other, and therefore the application of obtained results in the industrial conditions is considerably easier.

Fluent code used in the research has many multiphase models including VOF model. Calculations done with the use of VOF model are very time-consuming because obtaining the convergent solution requires making thousands of iterations. However, taking into consideration the level of reliability and adequacy of numerical simulations and accepted assumptions (correct choice of model and boundary conditions)

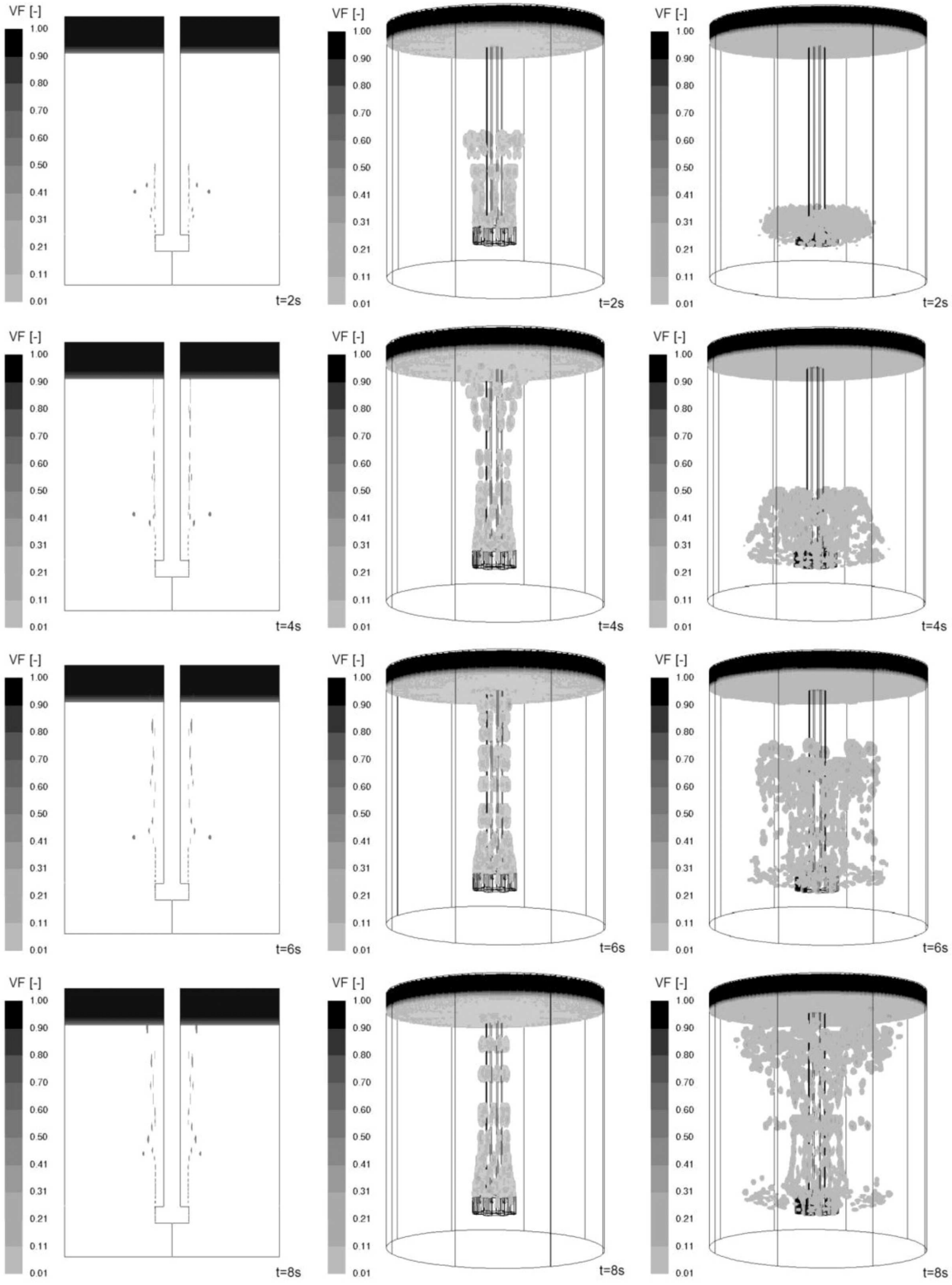

Fig. 3. Forecasted argon volume fraction inside the computational domain for analyzed test cases for various calculation time

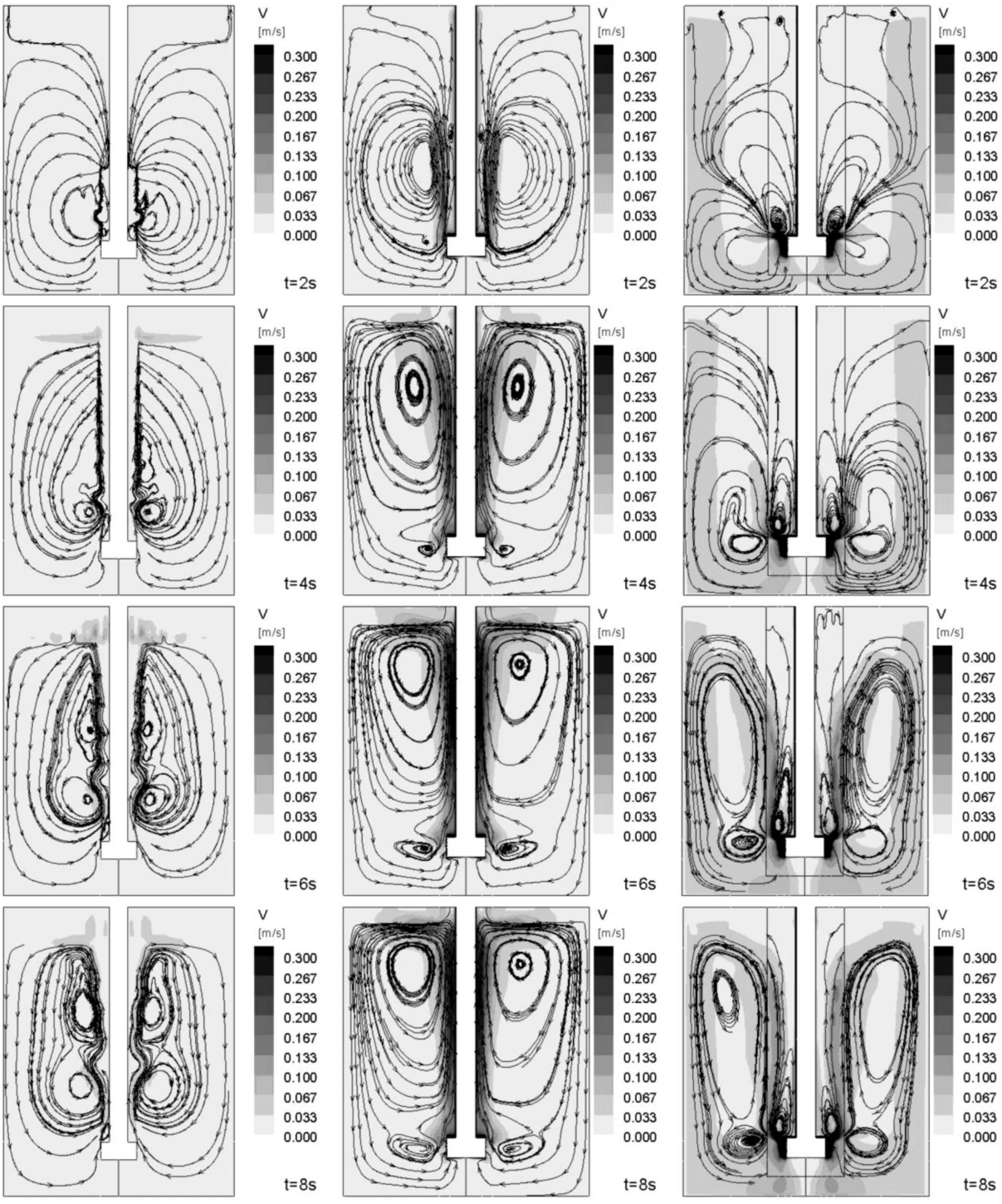

Fig. 4. Forecasted magnitude distribution for analyzed test cases for various calculation time

it can be claimed that results of presented computational simulations obtained with the use of this model can be applied to evaluate the phenomena occurring during the process of blowing argon into liquid aluminium. They also give helpful information about the velocity field and turbulence intensity occurring in the modelling agent; it is a valuable complementation of the conducted research programme.

This article presents results of the research for 2D case; it can be seen that the amount of gas bubbles in the schemes is rather small and does not match to the amount of gas bubbles obtained in real conditions (spatial view). Research concerning the possibilities of obtaining reliable results for 3D cases has been also done. Two different meshes were tried. First of them was stationary mesh with periodic boundary conditions. It gives better results than those obtained for 2D case; however there is still a problem with the distribution of gas bubbles in the whole refining reactor. Application of the movable mesh simulating the rotary movement of the impeller fully maps the

real object. Such calculations are very time-consuming but on the other hand results are more reliable. The aim of further research, which will be conducted in the nearest future, is to compare the results obtained from physical and numerical modelling for different flow rate of refining gas and rotary impeller speeds.

REFERENCES

[1] M. Saternus, J. Botor, Metalurgia **48**, 3, 175-179 (2004).
[2] M. Saternus, J. Botor, Archives of Metallurgy and Materials **55**, 2, 463-475 (2010).
[3] X.-G. Chen, F.-J. Klinkenberg, S. Engler, Light metals, TMS, 1215-1222 (1995).
[4] M. Saternus, T. Merder, P. Warzecha, Solid State Phenomena **176**, 1-10 (2011).
[5] L. Sowa, A. Bokota, Archives of Metallurgy and Materials **56**, 2, 359-366 (2011).
[6] A. Fornalczyk, S. Golak, M. Saternus, Mathematical Prob. in Engineering, Article ID 461085, doi:10.1155/2013/461085 (2013).
[7] K. Laber, H. Dyja, M. Kwapisz, Archives of Metallurgy and Materials **56**, 2, 385-392 (2011).
[8] A. Smalcerz, R. Przyłucki, Metalurgija **52**, 2, 223-226 (2013).
[9] B. Panic, Metalurgija **52**, 2, 177-180 (2013).
[10] F. Kerdouss, P. Proulx, J.F. Bilodesu, S. Vaudreuil, Light Metals TMS, 793-798 (2004).
[11] M. Warzecha, J. Jowsa, S. Garncarek, T. Merder, Iron and Steel Technology **1**, 11, 87-91 (2004).
[12] R. Gryboś, Podstawy mechaniki płynów – cz. II, PWN, Warszawa 1998.
[13] J.F. Wendt, Computational fluid dynamics, Springer-Verlag, Germany 1996.
[14] B.E. Launder, D.B. Spalding, Lectures in Mathematical Models of Turbulence, Academic Press, London 1972.
[15] M. Warzecha, Materials Science Forum **706-709**, 1556-1561 (2012).
[16] S. Golak, R. Zagorski, Metalurgija **52**, 2, 215-218 (2013).
[17] A. Smalcerz, Archives of Metallurgy and Materials **58**, 1, 203-209 (2013).
[18] ANSYSFluent, User's guide, version 13.0, Fluent Inc. 2013.

Received: 10 January 2014.