Publishing House
AKAPIT

COMPUTER METHODS IN MATERIALS SCIENCE
Informatyka w Technologii Materiałów

Vol. 10, 2010, No. 4

PREDICTIONS OF MELT CIRCULATION RATE IN A RH DEGASSER BY MATHEMATICAL MODELLING OF TWO PHASE FLOW

LEONARDO NEVES, CARLOS ANDRÉ RUY CARNEIRO, RAFAEL FERNANDES REIS, ROBERTO PARREIRAS TAVARES*

UFMG-Federal University of Minas Gerais
Avenida. Antônio Carlos, 6627 – Pampulha - 31.270-901 - Belo Horizonte - MG
Metallurgical and Materials Engineering Departament - Brazil
**Corresponding author: rtavares@demet.ufmg.br*

Abstract

Modeling of turbulent two phase flow is not a simple task, particularly under the conditions prevailing in metallurgical reactors. Different forces have to be considered and different models have been proposed to evaluate these forces. None of these models has been fully validated and they usually require the definition of coefficients whose values change according to the system being studied. In the case of the RH degasser, the variations of pressure and temperature inside the computational domain and the presence of a free surface add more difficulties to the development of an accurate mathematical model. In the present work, a mathematical model for two phase flow in a RH degasser has been developed using the commercial software Ansys-CFX. In this first approach, the mathematical model was developed to simulate two phase flow in a physical model of the RH degasser, using water to simulate steel. In this physical model, the variations of pressure are much less significant and an isothermal domain can be considered. Different models for turbulence and also for the interphase drag and non-drag forces have been considered. An Eulerian-Eulerian approach has been adopted. The predictions of the model in terms of melt circulation rate were compared to experimental results obtained in a physical model of a RH degasser. In this model, water was used to simulate liquid steel and air was used to simulate argon. The melt circulation rate in the physical model was determined by the injection of a solution of potassium chloride at the upleg snorkel and measuring the variation of its concentration with time at the downleg snorkel. Images taken from the physical model at the upleg snorkel and of the vacuum chamber were also used to validate the predictions of the model. The predictions of melt circulation rates with the different versions of the mathematical model were close to the experimental results, but none of the models could exactly reproduce the variation of the melt circulation rate with the gas flow rate. These results indicate that some adjustments are still required to improve the quality of the predictions of the mathematical model.

Key words: RH degasser, Two-phase flow, Mathematical modeling, Circulation rate, Drag and non-drag forces

1. INTRODUCTION

The RH process is a refining process used in the production of interstitial free steels. In this process, vacuum and inert gas injection promote the refining reactions (removal of C, H and N). The vacuum chamber is connected to the ladle containing liquid steel by two snorkels. The circulation of liquid steel between the vacuum chamber and the ladle is induced by gas injection through nozzles located at the upleg snorkel. The circulation rate has a significant effect on the decarburization and degassing rates and, consequently, on the productivity of the equipment.

 ISSN 1641-8581

Mathematical and/or physical modeling (Park et al., 2000; Park et al., 2001; Kamata et al., 1998; Seshadri & Costa, 1986) and also data from industrial units (Kondo et al., 1989) have been used to evaluate the melt circulation rate. The effects of argon flow rate, snorkels diameters, pressure in the vacuum chamber and location of the argon injection points, on the melt circulation rate have been studied.

Mathematical modeling of two phase flow in a RH degasser can be useful to understand the behavior and the interactions between the gas and the liquid. These interactions play a significant role in determining the efficiency of the process.

The first mathematical models (Kato et al., 1993) for fluid flow in RH degassers were single phase models focused on the flow pattern inside the ladle. In these models, the melt circulation rate was assigned based on empirical correlations, expressed in terms of gas flow rate, diameters of the snorkels and pressure in the vacuum chamber. These mathematical models could not predict the melt circulation rate and did not simulate the flow in the vacuum chamber, where most of the decarburization occurs.

More recently, Park et al. (2000; 2001) proposed a mathematical model to simulate fluid flow in a RH degasser. The effect of the gas on the flow was considered by introducing a buoyancy force in the momentum conservation equations. The trajectory and the shape of the plume were calculated by a separate model. The superposition of the plume zones was also included in the model. Expansion of the gas bubbles due to the pressure variations was considered and its effect incorporated in the buoyancy force. The predictions in terms of melt circulation showed good agreement with experimental results obtained in a water model; however, the mathematical model did not simulate the two phase flow, particularly the flows of liquid and gas inside the vacuum chamber, which present a significant effect on the decarburization rate.

In modeling the two phase flow in the RH process, different forces have to be considered and different models have been proposed to evaluate these forces. None of these models has been fully validated and they usually require the specification of coefficients whose values change according to the system being studied.

In the present work, a mathematical model for two-phase flow in a RH degasser has been developed using the commercial software Ansys-CFX. Different models for turbulence and for the inter-phase drag and non-drag forces have been considered. An Eulerian-Eulerian approach has been used.

Different simulations have been carried out analyzing the effects of the gas flow rate on the melt circulation rate. The results with the different versions of the model were also compared.

The predictions of the model in terms of melt circulation rate were compared to experimental results obtained in a physical model of a RH degasser. In this model, water was used to simulate liquid steel and air was used to simulate argon. The physical model was built in a 1:5 scale of an industrial RH. The similarity criterion was based on the modified Froude number. Images taken from the physical model at the upleg snorkel and of the vacuum chamber were also used to validate the predictions of the model.

2. METHODOLOGY

2.1. Mathematical modeling

The mathematical model was developed to simulate two phase flow in a physical model of a RH degasser. The physical model was built in a 1:5 scale of an industrial degasser. The main dimensions of the physical model and the characteristics of the gas injection system are presented in table 1.

The numerical simulations were developed considering steady-state condition. Fully converged solutions for the velocity fields (for water and air) and turbulence parameters were attained. Isothermal condition was assumed in all the simulations. The more accurate high resolution (Ansys, 2006) advection scheme, available in Ansys-CFX, was used in all simulations. After the convergence criterion has been met (RMS inferior to 10^{-6} for all variables), more iterations were performed to guarantee that the results were not significantly changing, particularly the volume fractions of air and water.

To solve the conservation equations for both phases, water (continuous phase) and air (dispersed phase), the commercial CFD software Ansys CFX was used. An Eulerian-Eulerian approach considering inhomogeneous multiphase flow was adopted. The free surface inside the vacuum chamber was considered flat. The conservation equations for this kind of simulation are well known and are not reproduced here. Under-relaxation was applied to all the conservation equations (physical time scale of 0.001 s).

Table 1. *Characteristics of the RH physical model.*

Parameters	Value
Ladle:	
- upper diameter (m)	0.720
- lower diameter (m)	0.648
- height (m)	0.750
- liquid level (m)	0.655
Vacuum chamber:	
- diameter (m)	0.415
- height (m)	0.700
- liquid level (m)	0.090
- pressure (Pa – gauge)	- 2700
Snorkels:	
- length (m):	0.312
- diameter (m)	0.120
- depth of immersion (m)	0.120
- distance between centers (m)	0.300
Gas injection:	
- flow rate (STP l/min)	50 – 500
- nozzles:	
○ number	10
○ diameter (mm)	1.0
○ position (below liquid level) (m)	0.04

Due to the small variations of pressure in the physical model, the expansion of the gas due to pressure variation was not considered. Expansion of the gas due to thermal and pressure effects must be included when simulating an industrial RH degasser. The buoyancy force was modeled considering the density difference between the two phases. The following interphase momentum transfer models were considered:

- drag force: Grace drag model (Ansys, 2006), with a volume fraction correction exponent of 2;
- turbulent dispersion force: model proposed by Lopez de Bertodano (Ansys, 2006), with a turbulent dispersion coefficient of 0.3.

The lift, virtual mass and wall lubrication forces were not considered in the present model. For the continuous phase, the standard k-ε model with scalable wall functions was adopted. For the dispersed phase, turbulence was modeled with the dispersed phase zero equation model (Ansys, 2006).

In flows with a dispersed phase, large bubbles tend to increase turbulence in the continuous phase due to the presence of wakes behind them. In the present model, the Sato Enhanced Eddy Viscosity model (Ansys, 2006) was used to simulate this effect. In the mathematical simulations, it was necessary to specify the diameter of the bubbles that are formed at the injection nozzles. Szekely and Themelis (1971) reported experimental results showing the variation of bubble diameters as a function of the orifice diameter and the Reynolds number. Based on these results, it was possible to estimate the bubble diameters for the conditions considered in the simulations. Values between 4 and 5 mm were determined. To verify if this parameter has a significant effect on the results, values between 2 and 5 mm were tested.

In numerical simulations of fluid flow, it is very important to guarantee that the solution is grid independent. To analyze the effects of the mesh distribution, different grid arrangements were considered. Grid independent solutions were obtained with approximately 1.3 x10^6 elements. Special care was taken to guarantee that the area for gas injection in the mathematical model corresponded exactly to that area in the physical model.

2.2. Boundary conditions

The entire RH degasser was simulated. Symmetry was not considered, since the model is intended to investigate the effects of clogging of some of the nozzles used for gas injection.

No slip condition was assumed at the solid walls and the scalable wall function option of Ansys-CFX was adopted. The free surfaces at the vacuum chamber and at the ladle were assumed flat and kept at a constant level. A zero shear stress condition was also assumed at the free surfaces. In the free surface at the vacuum chamber, a degassing boundary condition was also adopted.

At the gas inlets, a mass flow rate was specified. Mass flow rates corresponding to volumetric flow rates in the range of 50 to 500 STP l/min were simulated.

The numerical procedure was initiated assuming that the entire domain was occupied by water. A linear variation of the pressure along the vertical direction of the domain was also considered. That was crucial to promote convergence of the numerical procedure.

2.3. Experimental setup

A schematic view of the experimental set-up is depicted in figure 1. During the experiments, air supplied by a compressor was injected in the upleg snorkel. The flowrate was measured by a mass flowmeter and controlled manually. There were no individual measurements of flowrate for each injection nozzle. To equally distribute the gas flow rate among the nozzles, the air was first injected in the central region of a small chamber. This chamber was connected to each nozzle using pipes with the same length and diameter.

In the experiments, air instead of argon was injected in the upleg snorkel. Water was used to simulate liquid steel, since they both have approximately the same kinematic viscosity. Froude and modified Froude numbers were the same in the physical model and in the industrial installation.

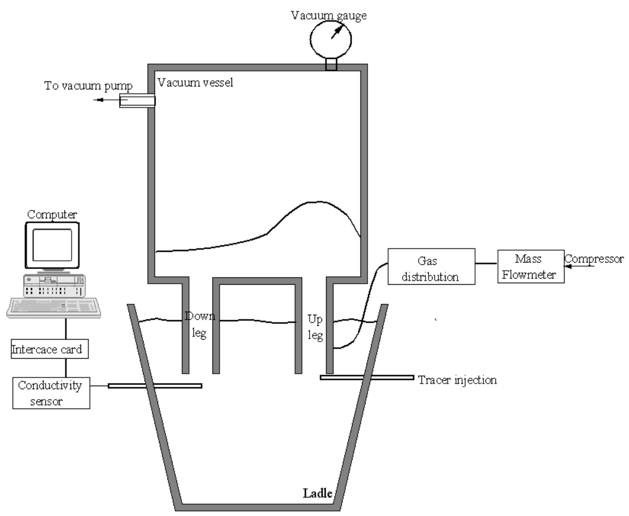

Fig. 1. *Schematic view of the experimental set-up*

The pressure in the vacuum chamber was controlled manually and monitored by a pressure gauge. The water levels in the ladle and in the vacuum chamber were controlled and kept the same in all the experiments.

The melt circulation rate was evaluated using the procedure adopted by Seshadri and Costa (1986). This procedure is very well established and is not reproduced here.

During the experiments, images of the cross section of the upleg snorkel were captured by a video camera. These images were compared to those obtained by post-processing the results of the mathematical model, identifying the area of the plume.

3. RESULTS AND DISCUSSION

3.1. Velocity field

Figure 2 presents the predictions of the water velocity field in the symmetry plane of the RH degasser (ladle and vacuum chamber). The flow pattern is very similar in all the three situations presented. Two recirculation regions are identified, one close to the jet coming from the downleg snorkel, and the other on the bottom of the ladle underneath the upleg snorkel. Figure 2 also includes an image obtained in the physical model, using the laser sheet technique. The flow pattern predicted by the mathematical model is very close to that observed in the physical model.

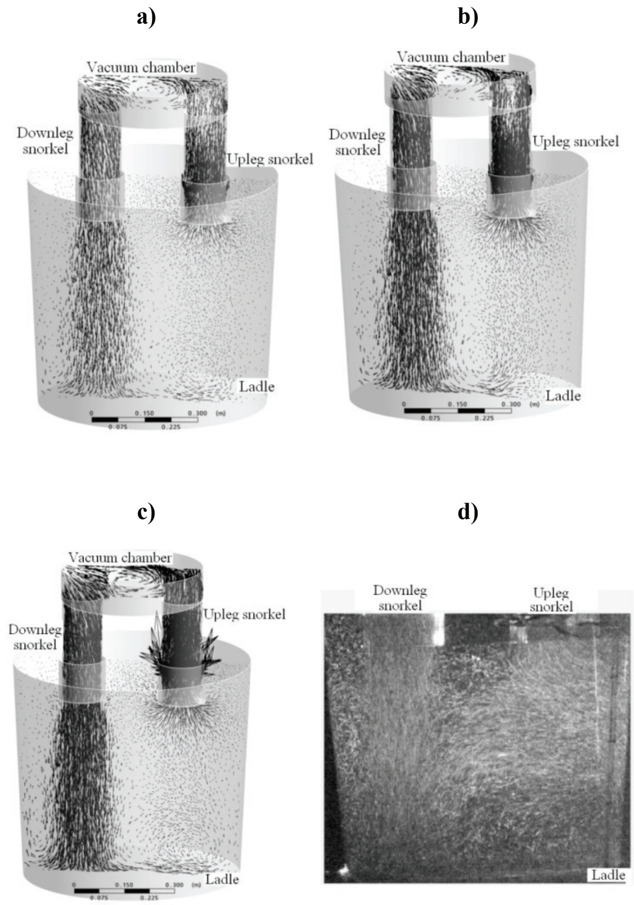

Fig. 2. *Predicted water velocity field for different gas flow rates and result of flow visualization experiment.*
a) Gas flow rate: 100 STP 1/min
b) Gas flow rate: 250 STP 1/min
c) Gas flow rate: 500 STP 1/min
d) Physical model. Gas flow rate: 250 STP 1/min. Flow visualization: laser sheet technique

3.2. Volume fractions

Figure 3 shows the predicted water volume fraction variations at the symmetry plane in the upleg snorkel for flow rates of 100 and 500 STP l/min. Figure 4 presents the variations of the water volume fraction at the cross section of the upleg snorkel 0.05 m above the gas injection nozzles. The penetration of the jets to the center of the upleg snorkel increases for higher flow rates; however, the penetration is small as compared to the images shown in figure 5, obtained in the physical model.

3.3. Melt circulation rate

The melt circulation rate was determined based on the predictions of the mathematical model, by evaluating the mass flow rate at the cross section of the downleg snorkel, using a predefined function of Ansys-CFX.

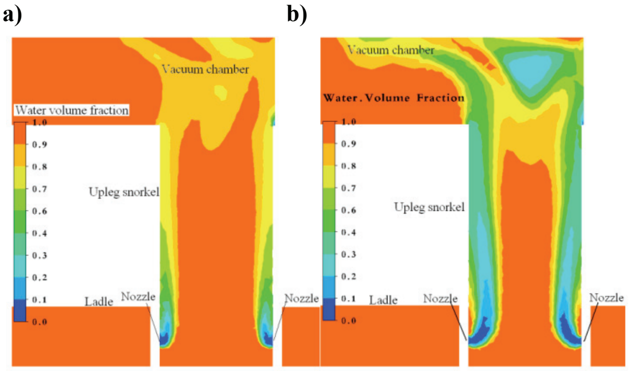

Fig. 3. *Water volume fraction at the symmetry plane of the upleg snorkel predicted by the mathematical model*
a) Gas flow rate: 100 STP 1/min
b) Gas flow rate: 500 STP 1/min

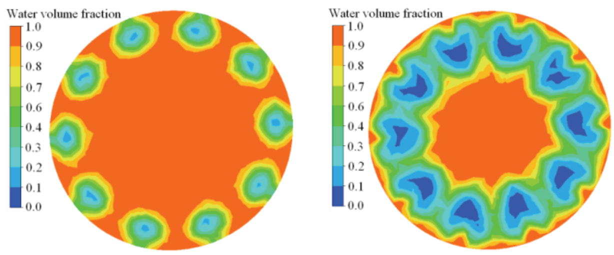

Fig. 4. *Water volume fraction at the cross section of the upleg snorkel 0.05 m above the gas injection nozzles, predicted by the mathematical model*
a) Gas flow rate: 100 STP 1/min
b) Gas flow rate: 500 STP 1/min

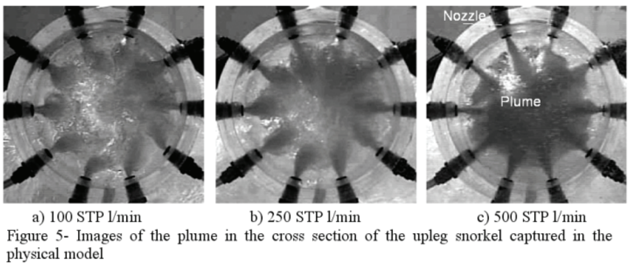

Fig. 5. *Images of the plume in the cross section of the upleg snorkel captured in the physical model*
a) 100 STP 1/min, b) 250 STP 1/min, c) 500 STP 1/min

As mentioned previously, preliminary simulations were performed considering different sizes for the bubbles formed at the gas injection nozzles. For bubbles with diameters ranging from 2 to 5 mm, the variations on the melt circulation rate were inferior than 5 %. Based on these results, it was decided to run all the simulations considering a bubble diameter of 5 mm, which is consistent with the experimental results reported by Szekely and Themelis (1971).

The variation of the circulation rate with the gas flow rate is presented in figure 6. Two curves are presented, one for the predictions of the mathematical model and the other for the results obtained in the experiments with the physical model.

The predictions of the mathematical model present a reasonably good agreement with the experimental results. Both curves indicate a tendency of the melt circulation to increase when the gas flow rate increases, but the predictions of the model indicate a more pronounced effect of the gas flow rate on the circulation rate. The reason for this difference was investigated by comparing the images of the plume in the upleg snorkel for different gas flow rates, as presented in figure 5. In this figure, the darker areas correspond to the regions mainly occupied by the gas. When the gas flow rate increases, the plume tends to concentrate in the central region of the snorkel.

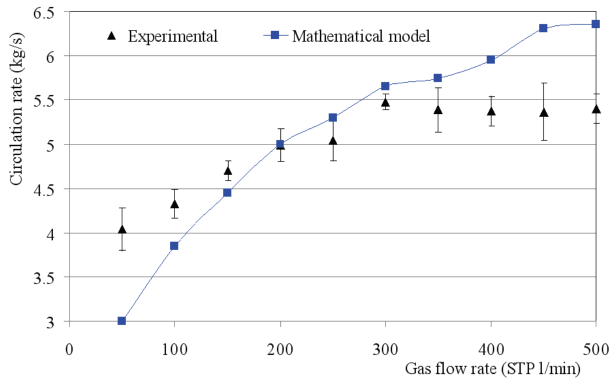

Fig. 6. *Comparison between the predicted and the experimental circulation rates*

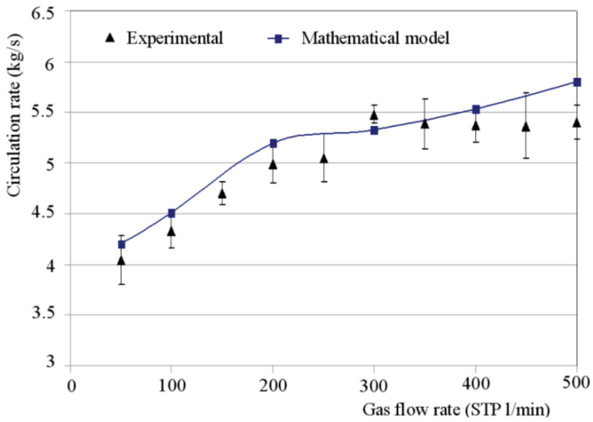

Fig. 7. *Comparison between the predicted and the experimental circulation rates, using the model proposed by Ishii and Zuber (Ansys, 2006)*

The jets coming from the nozzles interact with each other, forming a zone that is predominantly occupied by the gas. Due to this, the circulation rate tends to remain approximately constant or to show a slight tendency of reduction beyond a certain gas injection flow rate. At lower gas flow rates, this small penetration is responsible for the lower circulation rates predicted by the model. For higher gas flow rates, the strong coalescence of the jets in the central region of the snorkel, as seen in the images captured in the physical model, causes the stabilization of the circulation rate. The predictions of the mathematical model do not show a significant interaction of the jets. This explains the continuous increase in the circulation rate even for higher gas flow rates.

These results indicate that the current version of the mathematical model still requires improvements to be able to accurately reproduce the experimental data. To verify if other combinations of drag and non-drag forces, and models to express them, would give better results, several simulations combining these different forces were developed. Even the drag model proposed by Ishii and Zuber (Ansys, 2006), which is more suitable for flows with high volume fractions of the dispersed phase, was tested. This model considers the shape of the bubbles through the evaluation of the Eötvös dimensionless number. The results obtained when these changes were introduced are shown in figure 7. A slight improvement when compared to the results presented in figure 6 was achieved. One of the limitations of all these models is that they require the specification of so called *universal* constants. These constants are usually determined by matching the model predictions to the experimental data and they can vary in wide ranges depending on the system being studied. Most of these experimental data are restricted to bubbly flows in ducts, in conditions that are not very similar to those found in the RH process. Other combinations of models and *universal* constants are currently being tested to verify if the predictions of the model present a better agreement with the experimental results, particularly at the high gas flow rates. The introduction of breakup and coalescence of the bubbles might improve the results. This possibility will be investigated in future works.

4. SUMMARY AND CONCLUSIONS

A mathematical model to simulate the two-phase flow in a RH degasser has been developed and used to analyze the effects of gas flow rate on the rate. Different models for the drag and non-drag forces were tested.

The circulation rates predicted by the mathematical model presented reasonably good agreement with the experimental data, but the effect of the gas flow rate on the circulation rate was overestimated. The mathematical model did not reproduce the tendency of the circulation rate to level off at high gas flow rates.

The models used to evaluate the drag and non-drag forces require the specification of constants, which are determined by matching the model predictions to experimental data. The best results so far had been obtained with the Grace drag model, with a volume fraction correction exponent of 2, and with the turbulent dispersion force model of Lopez de Bertodano, with a turbulent dispersion coefficient of 0.3. The main difficulty in specifying these constants is that the values suggested in the literature are determined for flows in conditions that are not very similar to those found in the RH process. Other combinations of models and values for the constants are still being evaluated to improve the predictions of the melt circulation rate.

ACKNOWLEDGMENTS

The financial support of FAPEMIG – Fundação de Amparo à Pesquisa do Estado de Minas Gerais, Brazil - in the form of a research grant to R.P. Tavares, Process No. TEC - PPM-00197-09, is gratefully acknowledged.

REFERENCES

Kamata, C., Matsumura, H., Miyasaka, H., Hayashi, S., Ito, K., 1998, Cold model experiments on the circulation flow in RH reactor using a laser Doppler velocimeter, Steelmaking Conference Proceeding, Warrendale, *Iron and Steel Society*, 609-616.

Kato, Y., Nakato, H., Fujii, T., Ohmiya, S., Takatori, S., 1993, Fluid flow in ladle and its effect on decarburization rate in RH degasser, *ISIJ International*, 33, 1088-1094.

Kondo, H., Kameyama, K., Nishikawa, H., Hamagami, K., Fujji, T., 1989, Comprehensive refining process by the Q-BOP-RH route for producing ultra-low carbon steel, *Iron & Steelmaker*, 16, 34-38.

Multiphase Flow Modeling, 2006, *Ansys CFX-Solver Modeling Guide, Ansys*, Inc., Canonsburg, PA, 155-202.

Park, Y-G., Doo, W-C., Yi, K-W., Ahn, S-B, 2001, The effect of operating parameters and dimensions of the RH system on melt circulation using numerical calculations, *ISIJ International*, 41, 403-409.

Park, Y-G., Doo, W-C., Yi, K-W., Ahn, S-B., 2000, Numerical calculation of circulation flow rate in the degassing Rheinstahl-Heraeus process, *ISIJ International*, 40, 749-755.

Seshadri, V., Costa, S.L.S., 1986, Cold model studies of R.H. degassing process, *Transactions of ISIJ*, 26, 133-138.

Szekely, J., Themelis, N.J., 1971, *Rate Phenomena in Process Metallurgy*, John Wiley & Sons, New York, 687.

PRZEWIDYWANIE PRĘDKOŚCI CYRKULACJI CIEKŁEGO METALU W PROCESIE ODGAZOWANIA RH POPRZEZ MATEMATYCZNE MODELOWANIE PRZEPŁYWU DWUFAZOWEGO

Streszczenie

Modelowanie turbulentnego przepływu dwufazowego jest zadaniem trudnym, szczególnie w warunkach występujących w reaktorach metalurgicznych. W procesie tym należy uwzględnić działanie różnych sił, stąd w literaturze zostały zaproponowane różne modele do oceny tych sił. Żaden z tych modeli nie został w pełni zweryfikowany i modele te wymagają zazwyczaj zdefiniowania współczynników, których wartości zmieniają się w zależności od analizowanego systemu. W przypadku urządzenia do próżniowej rafinacji stali typu RH, zmiany ciśnienia i temperatury w obszarze rozwiązania oraz obecność swobodnych powierzchni są dodatkowym utrudnieniem dla zbudowania dokładnego modelu matematycznego. W niniejszej pracy przedstawiono matematyczny model dla dwufazowego przepływu w procesie odgazowania RH, stosując komercyjne oprogramowanie Ansys-CFX. W pierwszym przybliżeniu model został opracowany dla symulacji dwufazowego przepływu w fizycznym modelu urządzenia do próżniowej rafinacji stali typu RH, w którym stal zastąpiono wodą. W modelu fizycznym zmiany ciśnienia mają znacznie mniejsze znaczenie oraz dopuszczalne jest założenie warunków izotermicznych . Rozważono natomiast różne modele turbulencji i przyjęto brak sił międzyfazowych na granicy faz. W rozwiązaniu zastosowano sformułowanie Eulera. Przewidywania modelu w zakresie cyrkulacji cieczy zostały porównane z wynikami uzyskanymi z modelu fizycznego, w którym powietrze zastępowało argon. Prędkość cyrkulacji w modelu fizycznym została wyznaczona przez wstrzyknięcie roztworu chlorku potasu przez króciec wlotowy i pomiar zmian stężenia tego związku przy króciec wylotowy. Obraz przepływu uzyskany z fizycznego modelu oraz w komorze próżniowej zostały wykorzystane do weryfikacji modelu matematycznego. Przewidywane prędkości cyrkulacji dla różnych wersji modelu matematycznego były zgodne z danymi doświadczalnymi, ale żaden z modeli nie potrafił odtworzyć dokładnie zmian prędkości cyrkulacji powodowanych przez zmiany prędkości przepływ gazu. Uzyskane wyniki wykazują, że dalsze korekty są potrzebne aby poprawić jakość przewidywań modelu matematycznego.

Received: September 14, 2010
Received in a revised form: October 11, 2010
Accepted: October 18, 2010