Mathematical Modelling of Steel Flow at Kambara Reactor

Marcelo Kruger[1], Ricardo Barros[1], Regis Ataídes[1], Marcus V. F. dos Reis[1], Bruno Augusto Batista Ribeiro[2], Clóvis R. Maliska[3]

[1] ESSS – Engineering Simulation and Scientific Software
[2] Magnesita Refratários S.A.
[3] UFSC – Universidade Federal de Santa Catarina

ABSTRACT

The KR Reactor has been used as one of the processes in the treatment of hot metal desulphurization due to its efficiency and low operation cost. The process consists of creating an emulsification of a basic slag through the agitation energy promoted by a rotating impeller which is immersed in the metal. In order to evaluate the influence of the geometric and operational parameters in the mixture efficiency, a computational model of Reactor KR has been developed. The employed methodology consisted of solving the fluid flow equations numerically using the computational package ANSYS CFX® on the KR reactor model. Variations in the position, depth and rotation of the impeller have been evaluated. The results of velocity field and homogenization time have been evaluated.

1. INTRODUCTION

Due to the competitiveness and the requirement for high quality products, it is becoming necessary a deeper knowledge of the phenomena that are involved on the steelmaking processes.

The hot metal desulphurization in ladles has been gaining importance, once it presents good efficiency with a low operation cost. The process, shown in Figure 1, consists of the reaction of the basic slag with the hot metal through the agitation energy promoted by a rotating impeller which is immersed in the bath. Then, knowing the flow features of the process has become essential.

Based on this scenario, the goal of this work is to develop and implement a computational model that could be able to represent the flow inside a Kambara Reactor and then evaluate the influence of geometric and operational parameters on the Reactor's efficiency.

In a first step, the steady-state and a single phase model assumption was considered. Two different models of turbulence have been evaluated. The numerical results were compared with experimental data and the agreement between them was very close. Even with these results, a new model considering multiphase flow has been developed to prove the simplifications that were made when working with single phase.

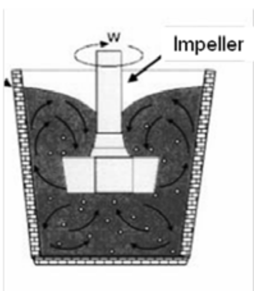

Fig. 1: Kambara Reactor

Thus, after the validation stage is concluded, some geometric and operational parameters have been evaluated. The Reactor's efficiency has been analyzed by the mixture homogenization time. A tracer is injected as a passive scalar variable on CFD model and its concentration is monitored during time in several points inside the ladle.

2. METHODOLOGY

2.1 Computational Mesh

The numerical model for fluid dynamics simulations must describe the most relevant aspects of the real physical problem. The geometric domain of this work is divided in two regions: rotational domain and stationary domain. To model the flow inside this domain, Navier-Stokes equations have been used.

To apply the equations on the model, the geometry has been discretized with a hybrid computational mesh, which contains around 500 thousand nodes and is made of tetrahedrons and prisms.

The Figure 2 shows a cut plane on the domain with the mesh.

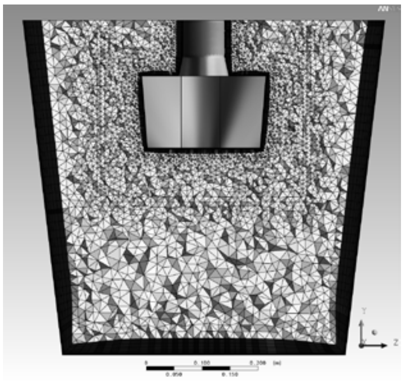

Fig. 2:Computational Mesh

2.2 Numerical Model

As mentioned in the previous section, two stages have been developed. In the first one, a single and a multi phase (air and water) model was implemented in order to validate the numerical model. The validation was done comparing the free surface elevation on top of the ladle with experimental data. It was evaluated for 6 values of impeller rotation.

The walls of the domain, including the impeller, were considered without slip, except on the top, where was applied the free slip condition for the single phase model and the opening condition, for the multi phase model, once it would be considered that air comes out of the domain.

In the second stage, it was used the single phase model to evaluate the influence of geometric and operational parameters on the mixture homogenization time. The parameters evaluated are shown on the table 1 and illustrated on the Figure 3.

Impeller Rotation (rpm)	Impeller immersion level (mm)	Impeller eccentricity (mm)
80	1400	Concentric
90	1600	Eccentric
110	1800	Eccentric
130	2000	

Tab. 1: Geometric and operational parameters

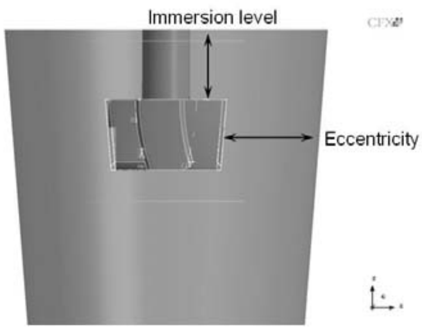

Fig. 3:Geometric parameters

The following assumptions have been considered**:**

- Steady-state flow;
- Turbulent flow - SST model;

Once the results of the steady state flow were achieved, the tracer was injected at the top considering a transient regime. Then, the homogenization time is evaluated monitoring the tracer concentration in the interior of the ladle.

3. RESULTS

Initially, the results compare the free surface elevation on top of the ladle between numerical and experimental data. The pressure on top of the ladle was calculated to evaluate the free surface elevation on the single phase model according to the following expression:

$$Elevation = \frac{\text{Pr}essure - Mean\,\text{Pr}essure}{\rho .g} \qquad (1)$$

On the multiphase model, an isosurface of water volume fraction was calculated. The Figure 4 shows the free surface elevation using the equation (1) on the single phase model and the isosurface of water volume fraction of 0.5 on the multi phase.

It is possible to verify that the results are very close.

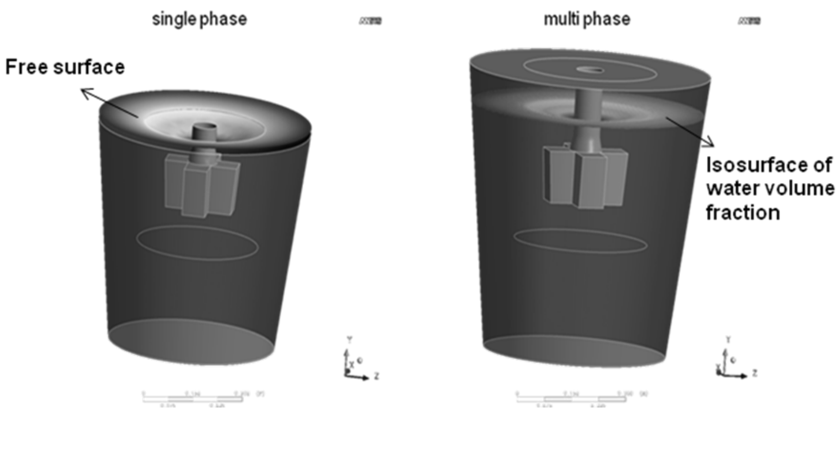

Fig. 4:Free surface elevation

Finally, to check and validate the methodology, the results were compared with experimental data and the agreement between them was very close, even with the simplifications on the single phase model. The Figure 5 shows the results computed with the single and multiphase flow and with *SST* and *K-Epsilon (KE)* turbulence models.

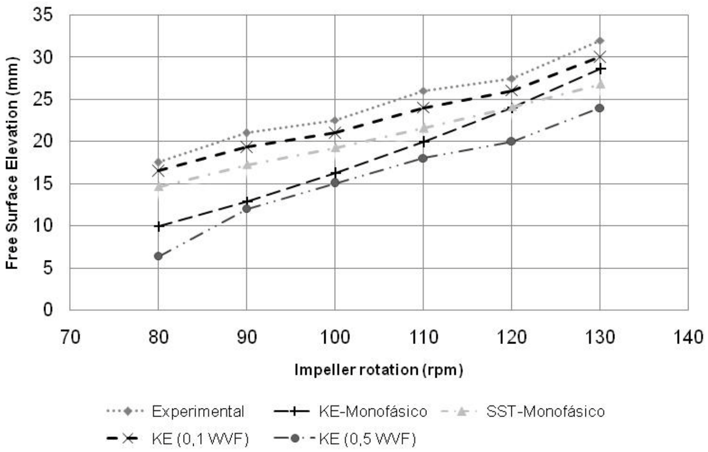

Fig. 5:Compare between numerical and experimental data

Once the numerical model is validated, some geometric and operational parameters were evaluated as previously mentioned.

At Figure 6, velocity contour in a cut plane is shown for each configuration considering the impeller rotation of 110 rpm.

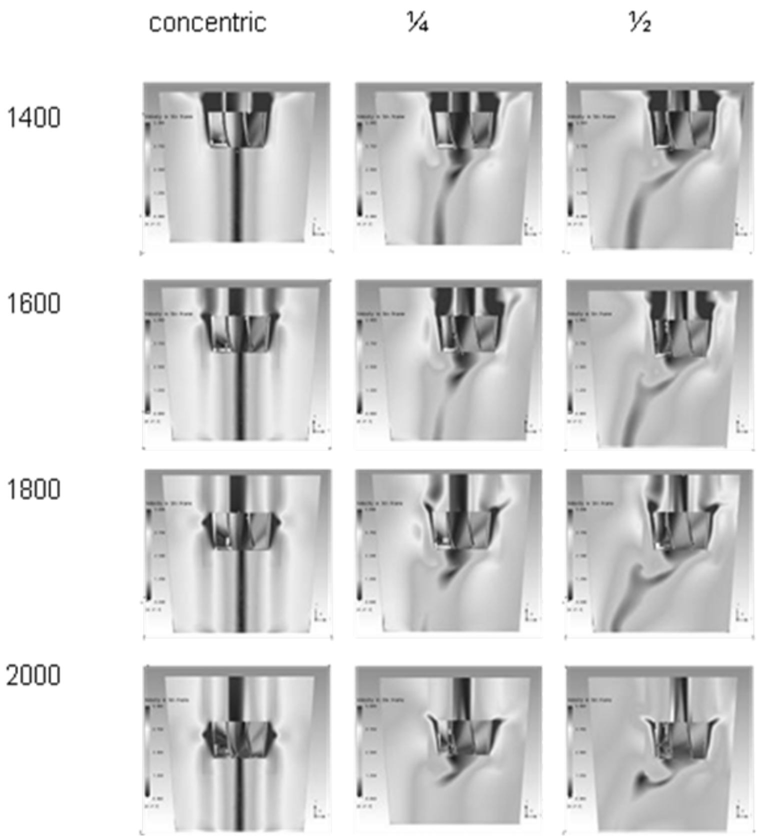

Fig. 6:Cut plane of velocity

It is possible to verify that there is a central vortex on the concentric cases, resulting in a “solid” rotation on the Reactor. The eccentricities promote a decrease on values of velocity and a displacement on the central vortex.

The impeller immersion level showed importance since increasing the depth of immersion reduced the intensity of the vortices.

To evaluate the mixture homogenization time, or, the time that tracer concentration is uniform in all the ladle, the variance of tracer concentration was calculated at every time step and then a curve of Time x Variance of tracer concentration was created. A regression at the curve was applied and the coefficients, a and b were extracted. Thus, the homogenization time was calculated as:

$$T = \left(\frac{V}{a}\right)^{\frac{1}{b}} \tag{2}$$

where *T* is the time and *V* the variance.

Considering the mixture uniform for a variance of 1.0e-6, the Figure 7 shows the calculus for each configuration.

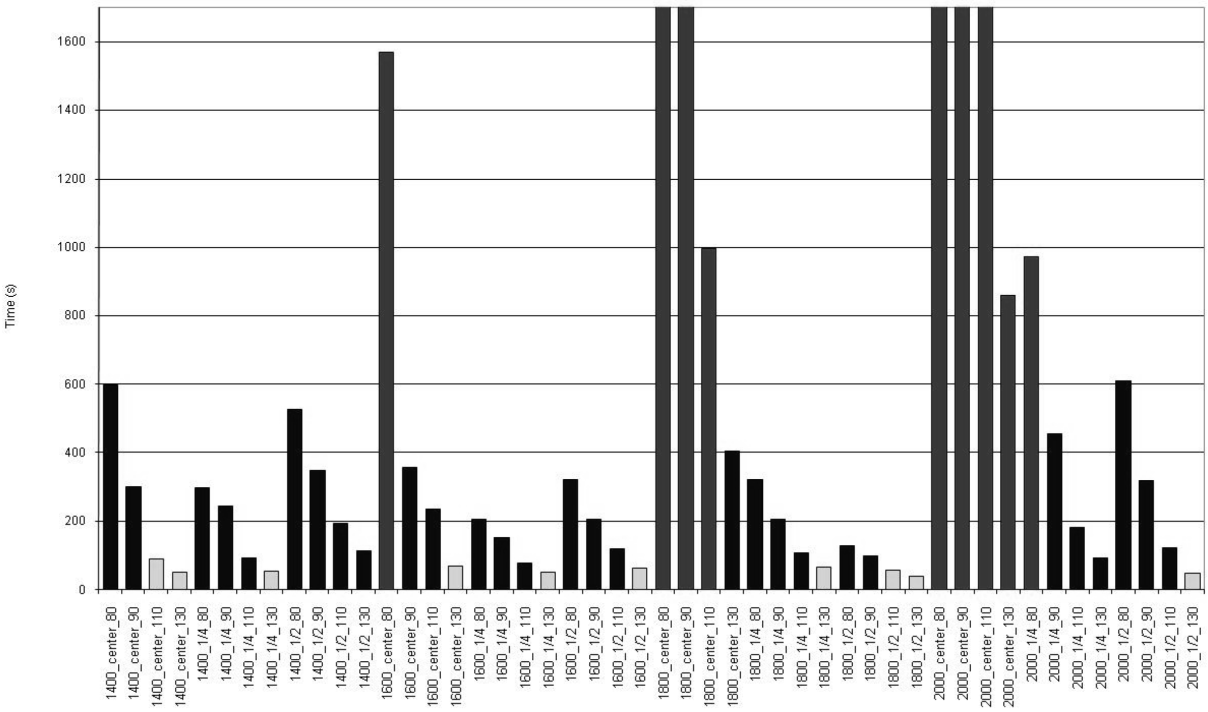

Fig. 7:Homogenization time

The results of tracer indicated that:

- Increasing rotating velocity reduces the mixture homogenization time;
- Increasing the impeller eccentricity decreases the mixture homogenization time;
- concentric configurations, and increasing impeller immersion increases the mixture homogenization time;

4. FINAL CONSIDERATIONS

The development of the numerical model allows for a better understanding of the flow behavior inside the KR reactor.

Once the numerical model is validated, it could give useful insights to engineers to make important decisions concerning the improvement of the desulphurization process.

5. REFERENCES

[1] C. R. MALISKA - Transferência de Calor e Mecânica dos Fluidos Computacional, Rio de Janeiro, RJ, Brasil: ed. LTC - Livros Técnicos e Científicos Editora S. A., 1995.

[2] ANSYS CFX - Documentation.

[3] F. R. MENTER – Two-equation eddy viscosity turbulence models for engineering applications, AIAA-Journal, 32(8), pp. 1598 – 1605, 1994.