Mathematical Modelling of Gas Injection in a Continuous Casting Tundish

Marcelo Kruger[1], Regis Ataídes[1], Marcus V. F. dos Reis[1], Rodrigo N. Borges[2], Clóvis R. Maliska[3], Carlos Alberto Silva[4]

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

ABSTRACT

The practice of gas injection inside a continuous casting tundish has gained special attention because it can eliminate the usage of dams and weirs. In addition, the generation of small bubbles also increases the interface area between bubble and liquid metal, promoting the collision and trapping of inclusions [ZHANG, 2004]. In order to get a better knowledge of the process, a computational model of gas injection inside a tundish has been developed and validated with experimental data. The methodology employed consisted in solving numerically the equations of water flow inside the tundish using the computational package ANSYS CFX® by finite volume methodology. A computational model of turbulent and multiphase flow considering gas injection inside the tundish has been built. The model has been validated through a comparison of Residence Time Distribution (RTD) and inclusion removal with experimental measurements. The results presented very good agreement with experimental data. Once the computational model is validated, it can be used for implementing new geometric configurations and/or changing operational parameters.

1. INTRODUCTION

In recent decades the usage of computational tools has become increasingly crucial in solving engineering problems, due the greatest flexibility and agility in relation to conventional methods.

The development of these tools has allowed the analysis of engineering problems very close to the real operation conditions.

In metallurgical industry, where the conduct of experiments with the real operation conditions is often impossible to implement due to high temperatures, the risk of an explosion, etc., the usage of computational tools has gained further importance.

The continuous casting tundish, as showed in Figure 1, is the last of various processes in which the metal is submitted in the continuous casting and one of the most efficient places to promote inclusions flotation and thus raise the quality of final product. Then, know and optimize the flow features inside the tundish 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 tundish considering the gas injection. A computational model of turbulent and multiphase flow has been built and validated through a comparison of Residence Time Distribution (RTD) and inclusion removal with experimental measurements.

To evaluate the inclusion flotation the Lagrange modeling has been used. This method integrates the three-dimensional trajectories of the particles based on the forces acting on them from surrounding fluid and other sources. Turbulence is usually taken into account by a random motion, superimposed on the trajectory.

The gas has been injected just in one side of the tundish, allowing comparison of injection efficient in inclusion flotation.

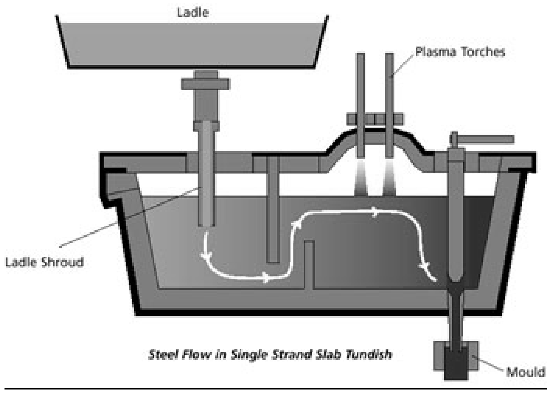

Fig. 1: Tundish schematic view

The Residence Time Distribution curve has been obtained with a tracer that is injected as a passive scalar variable on CFD model and your concentration is monitored during the time in the strands of tundish.

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 composed by the tundish, the ladle shroud, the plug of gas injection. For the modeling of 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, containing about 600 thousand nodes, between tetrahedrons and prisms.

A cut plane on the domain with the mesh and a zoom on the central region, with ladle shroud and the turbulence stop, is shown in Figure 2.

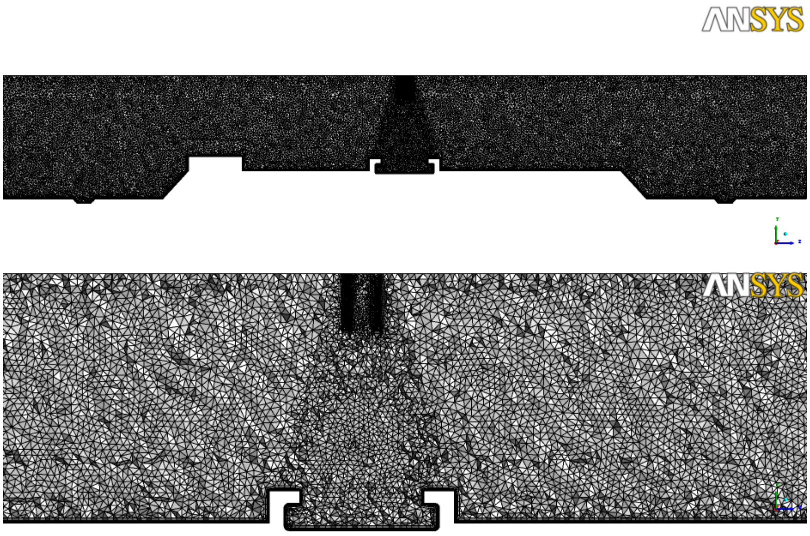

Fig. 2:Computational Mesh

2.2 Numerical Model

As mentioned in the previous section, two stages have been developed. In the first a multi-phase (air and water) model has been implemented, in order to validate the numerical model. The validation has been performed comparing RTD with experimental data.

The walls of the domain were considered as no slip condition, except on the top, where an opening condition was applied, once it would be considered air coming out of the domain.

In the second stage, a Lagrange model has been applied, in order to evaluate inclusion flotation.

For the each case two configurations have been evaluated, as shown on table 1 and illustrated on Figure 3.

Table1 : Configurations evaluated

Plug Position	Gas mass flow (l/min)
External	10
Internal	10

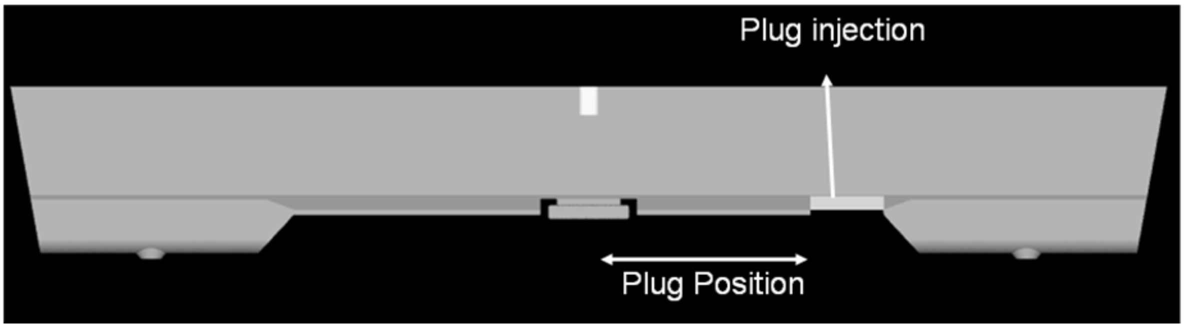

Fig. 3:Geometric parameters evaluated

The following assumptions have been considered**:**

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

With the results of the steady state flow the tracer was injected at the ladle shroud considering a transient regime. Then, the RTD is evaluated monitoring the tracer concentration in the strands.

3. RESULTS

First of all, the results compare the RTD between numerical and experimental data. As mentioned in the previous section, the curves have been plotted with a tracer, either in experimental and numerical models.

In the Figure 4 the curves between experimental and numerical model are compared. The curves are plotted to both strands, with and without gas injection, for the external plug injection case.

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

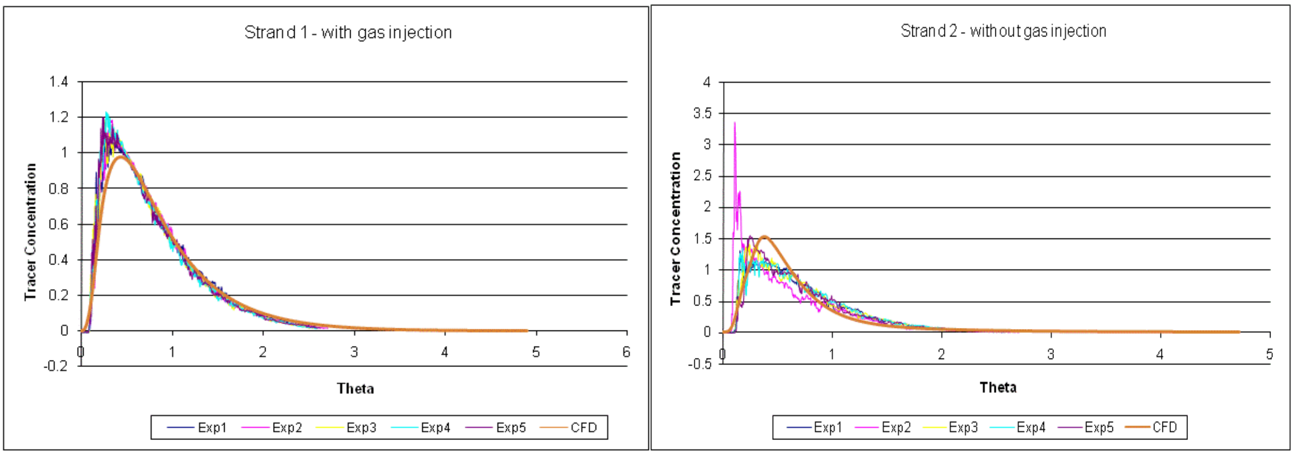

Fig. 4:Residence Time Distribution for external plug case

In the Figure 5 the same results but now for curves of internal plug injection case are shown.

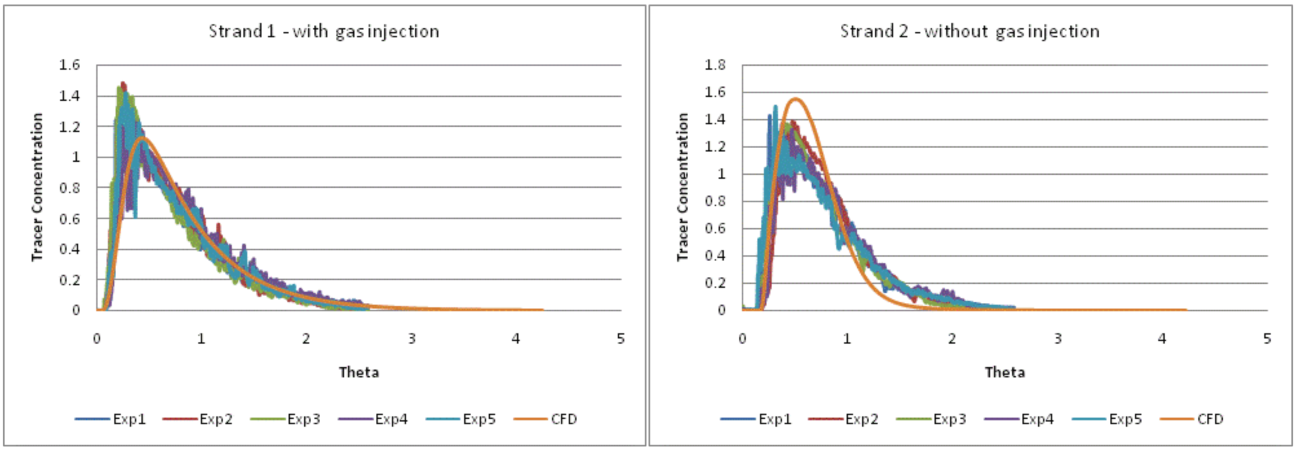

Fig. 5: Residence Time Distribution for internal plug case

The curtain of air that is created with the injection is shown in Figure 6. The same behavior could be observed on the experimental model.

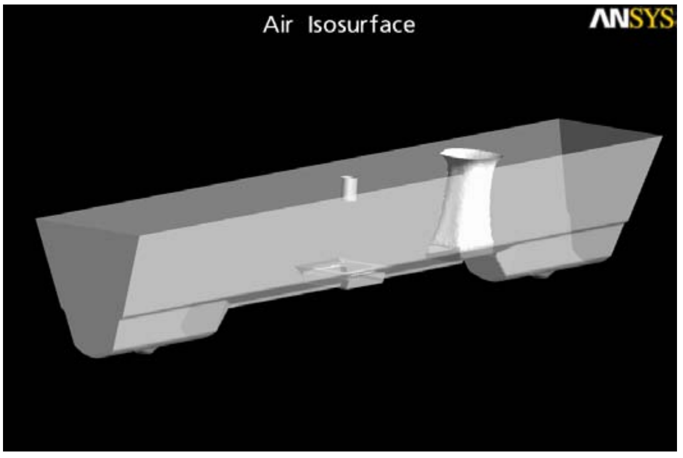

Fig. 6: Air Isosurface

Once numerical model showed satisfactory validation, the second stage was the development of a Lagrange model and the verification of the gas injection influence on the inclusion flotation.

One of simplifications on this method is that the equations of motion of individual particles that are solved without considering collisions between particles.

A restitution coefficient of 0 on the top of the tundish has been used. Thus, every inclusion that touches the top is captured. Inclusions of 50 micron, equivalent to the experimental particles, have been analyzed.

In the Table 2 values collected on the top and on the strands for both cases, external and internal plug position, are presented.

Table 2: Percentage of inclusions collected on the top and on the strands

	Inclusion Collected - External Plug Case		
	Inclusion 50 micron		
	Collected Top %	Collected Strand 1 %	Collected Strand 2 %
Exp1	62.65	6.02	31.33
Exp2	61.54	6.15	32.31
Exp3	63.64	6.06	30.3
Exp4	63.41	6.1	30.49
Average	62.81	6.08	31.11
CFD	75	1	13

	Inclusion Collected - Internal Plug Case		
	Inclusion 50 micron		
	Collected Top %	Collected Strand 1 %	Collected Strand 2 %
Exp1	63.87	5.04	31.09
Exp2	66.35	4.81	28.85
Exp3	66.32	5.26	28.42
Exp4	67.05	4.55	28.41
Exp5	68.22	4.65	27.13
Average	66.84	4.69	28.47
CFD	76	1.5	12

The results show some quantitative differences that may be related to the approximations used in the Lagrange model.

Meanwhile, a qualitative similarity in relation to removal of each strand and the top can be verified. Both, experimental and numerical results, show an efficient removal of particles in the strand where the gas is injected. The upward flow generated by the "curtain" of gas increases considerably the inclusions removal.

Another factor that can be observed in both models is that, the change of the plug position does not produce a big influence on the removal efficiency.

4. FINAL CONSIDERATIONS

The development of a numerical model allows a better understanding of the flow behavior inside the tundish with gas injection.

Once the numerical model is validated with experimental data it could give useful insights to engineers to make important decisions concerning the improvement of 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.

[4] L. ZHANG, J. AOKI, B. G. THOMAS - Inclusion Removal by Bubble Flotation in Continuous Casting Mold, MS&T 2004, Conference Proceedings – 161.