

Computer Simulation of Fluid Flow Dynamics in a Novel Design of a Continuous Casting Tundish

Gerardo Barrera C*, Ramiro Escudero G., Armando López M.

Institute of Research in Metallurgy and Materials. Universidad Michoacana de San Nicolas de Hidalgo, Ciudad Universitaria Edificio "U", Apartado postal 888, C.P. 58000, Morelia, Michoacán, México.

Abstract: The main role of the tundish is to provide and distribute the molten steel, at a constant rate to the casting mold; however, because of the most rigorous quality requirements in recent years, the tundish has acquired great importance in eliminating inclusions. Traditionally, the continuous casting tundish is related to a rectangular container. In this research work the fluid dynamic of an innovative design of a continuous casting tundish consisting of a cylindrical body with half spherical ends was studied by using mathematical modeling. The dynamic behavior of the fluid in the new design showed that the curvature of the physical model will decrease the impact of the steel jet in the inlet and the fluid will tend to rise evenly through the walls of the tundish, since the spherical caps are regions of low velocity, in which much of the dead volume of the system is contained, thus it causes the phenomenon of vortex disappears. This behavior was compared with that of the straight type tundish. From experimental results, the piston volume was greater in the new tundish design as compared with the straight type, this is a clear evidence of the inhibitory effect of the concave shape of the proposed one. Base on this results, it was reduced the lengthwise size of the tundish, resulting a reduction of a dead fraction volume of around 15%.

Keywords: Continuous casting, Mathematical simulation, steel tundish, cylindrical tundish.

I. Introduction

In the steelmaking industry, due to the rising steel quality requirements emphasizing on reducing production costs, the tundish has evolved into a steel refiner, as it is the last stage of the steel before being solidified. Currently designing a tundish has been aimed at enabling to carry out various metallurgical operations such as separation and flotation of inclusions as well as the thermal and chemical composition control of the steel ^[1].

To accomplish the efficiency of this process, a controlled fluid flow within the tundish is required; otherwise, can impair the quality of the steel transported to the ladle. In recent years the general tendency for controlling fluid flow in the tundish has been the placement of flow modifiers devices, which are artifacts of simple geometry, made of refractory material resistant to high temperatures. The main flow modifiers are devices used as turbulence inhibitors, air curtains and dams ^[2].

Due to the high operating temperatures in the tundish (around 1600 ° C) and the opacity of steel, coupled with the technical difficulties of testing a prototype in plant, it is very difficult to study the behavior of the system directly. For these reason new techniques of study, such as mathematical and physical simulation, can offer great results with a high grade of confidence.

In this research work by means of mathematical modeling, the study of the fluid dynamics of a newfangled tundish design of the continuous casting machine has been carried out, it has a cylindrical geometry with hemispherical ends, and without flow modifiers inside. Theoretically this could improve the geometry of the tundish fluid flow and thus metallurgical operations carried out at the tundish would be more efficient.

II. Mathematical Simulation

Computer fluid dynamic simulation is an excellent alternative to study the behavior of fluid flow in detail in continuous casting tundish. This technique is based on the numerical solution of the governing equations of the system by means of numerical methods based on the use of finite volume or finite difference algorithms.

The fluid flow in the continuous casting tundish is extremely complex, since it has a high turbulence region, located in the entrance area, and in the rest of the tundish can be considered as a transition zone. In the mathematical modeling of the turbulent flow, characterized by high velocity gradients rates, the use of a set of equations, called turbulence models, is necessary. Unfortunately, there is no turbulence model applicable to all in-plant situations due to the choice of the turbulence model will depend on: the phenomena involved in the system, the required level of accuracy, and the available computational resources. In the study of the fluid flow, the most used model by researchers in recent years is the $k - \epsilon$ standard model.

The software used in this research was Fluent[®], which uses a technique called finite volume discretization. This technique has the advantage that it can be adapted to any type of meshing, either structured or unstructured. In this investigation, first was carried out the drawing of the geometry, the meshing was made by using the preprocessor Gambit[®]. Also in this same package system boundary conditions are assigned. Once it is done, the file is exported to Fluent[®], where the governing equations of the system, the working fluid, water in this case, and other parameters are selected. Once the set of equations are solved, at steady state, the performance of the fluid is analyzed using velocity vectors and contour. To obtain the residence time distribution curves (RTD) in Fluent[®] it is necessary to simulate the system under an unstable state by injecting a virtual tracer and monitoring the changes in concentration at the entrance and exit points of the tundish^[3,4].

The fluid flow in the continuous casting distributor is extremely complex^[5, 6, 7, 8], since it is a region of high turbulence zone located at the inlet and at the outputs of the tundish, which corresponds to a transition zone. In the transition zone the effects of laminar and turbulent flow are equally important, whereas in the turbulent areas, the strictly laminar effects are negligible.

2.1 Computational Fluid dynamics (CFD).

It is an engineering tool to simulate in detail any system or equipment in which there is involved fluid flow, heat or mass transfer phenomena. It is based on the numerical solution of the fundamental equations of conservation of mass, energy and momentum in a domain (geometry) specific discretized, i.e. that is converted into a grid of points (volumes or finite elements). As a result, the values of all system variables (pressure, velocity, temperature, composition, etc.) at each grid point and computing as a function of time in transient processes are obtained. The CFD computational steps are:

I. Problem identification and preprocessing.

- Drawing of the specific geometry to be simulated,
- Generation of the mesh.,
- Definition of the physical aspects of the problem (components present and phenomena to study.),
- Definition of the initial and boundary conditions.

II. Numerical solution of the Problem.

- Defining the parameters of the numerical calculation.
- Selecting the method of solution

Post processing

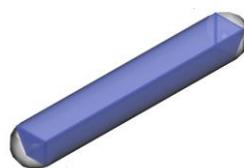
- Results analysis and generation of graphics, images or videos thereof

The grid or mesh systems are classified as structured, unstructured and hybrid. The structured grids are formed by lines, forming a curvilinear coordinate system. Thus the cells are rectangular in 2D and hexahedral in 3D. The unstructured grids are composed of lines that do not form a curvilinear coordinate system. Their cells may be triangular or tetrahedral or a combination of several forms. Hybrid grids involve a combination of structured and unstructured grids.

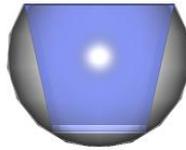
III. Experimental development

The proposed tundish can be defined as a cylinder with hemispherical caps at the ends, based on the straight type tundish to make the changes. In the first stage of the experimentation a comparative was made between the geometrical concepts of the proposed one with the original. Figures 1 (a) and (b) show a comparison between the proposed tundish and the straight type, it should be noted that both have the same height and length. When comparing the two types of tundish there is a clear increase in the capacity of the proposed one, since the distributor straight type has a capacity of 40 tons, whereas the capacity of the proposed is about 60 tons.

Some of the objectives of mathematical modeling include, first visualize the flow behavior as well as the study the fluid dynamics of the liquid steel in the new design and secondly to get an idea of the performance of some variables such as the cooling of steel, inclusions flotation, the installation of devices in the tundish, etc. It is always recommended, in first place, to carry out the mathematical modeling, as it saves us hours of physical modeling and unnecessary changes in the real tundish. The parameters obtained in this stage are validated by the technique of physical modeling, i.e. the scale model of the proposed tundish. This experimental technique saves time, money and effort.



(a)



(b)

Figure 1.- Proposed modification of the tundish with spherical taps at the ends; (a) isometric and (b) lateral views.

3.1 Mathematical modeling.

Mathematical modeling is the part of the experimental procedure, aided by hardware and software, with the main objective to solve numerically the governing equations of the fluid flow for the fluid dynamic behavior of the system. The software used was the preprocessor Gambit[®] (Geometry and Mesh Building Interactive Tool) and the processor Fluent[®]. This is done in the following steps: geometry drawing, mesh generation, establishment of border conditions, simulation conditions and the numerical solution.

3.1.1 Geometry Drawing.

At this stage the design is done, i.e., the geometry of the proposed tundish is drawn in the preprocessor Gambit[®] and then the mesh of the model is generated. The geometry of the system not only involves drawing the tundish but also the inputs and outputs of the nozzle, in addition to the geometric cuts. The cuts relate to the splitting of the geometry of the system, having as objectives, separate two objects whose volumes are spliced as in the case of the tundish and the nozzle inlet, besides the fact that this may have a less distorted mesh.

3.1.2 Mesh Generation.

Meshing is the discretization of the system in this case using control volumes. In this work a hybrid mesh was used, i.e. the combination of structured and unstructured meshing. This last one is used in the proposed tundish of hemispherical ends, due to the complexity of the geometry and that implementation of cuts in this area is not feasible. Figure 2 shows the area of the unstructured meshing present in the system on a longitudinal plane of the tundish.

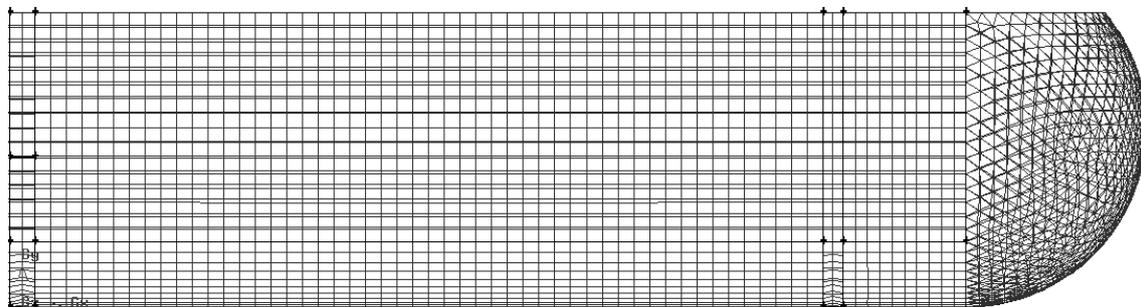


Figure 2.- Hybrid meshing of the central longitudinal plane showing in the hemispherical cap the non structured mesh.

3.1.3 Considerations and Border Conditions.

The boundary conditions specify the values of the variables at the boundaries of the physical model. The governing equations are similar to many of the problems of fluid flow, so the boundary conditions are those that define the problem and difference from others. To set appropriate considerations are important because they help to simplify the model.

The boundary conditions and assumptions used were as follows:

- No sliding flow on all solid surfaces. This condition specifies that the velocity of the fluid in contact with a solid surface is the same as that of said surface. For a stationary container, this means that the speed becomes zero in the stationary surfaces, such as the tundish,
- The entry, of the fluid flow, in the tundish was defined as "speed input", i.e. in units of m/s. This is due that, if a mass flow entry will be selected, it will require an excessive fine mesh in the input plane, in order to take the appropriate speed value. The inlet velocity was calculated from the Froude similarity criterion and is equal to 0.55 m / sec,

- Symmetry Boundary Condition. This condition is very useful to reduce the computational requirement and basically implies no flow on the axis of symmetry. This allows studying the model in a fraction of the model, i.e., in a fourth of the model for example,
- The fluid within the distributor behaves like a Newtonian fluid.
- The fluid is incompressible, which means that the density of the fluid does not change over time, pressure or temperature. This assumption is not true for the liquid steel but is necessary for mathematical modeling because it assumes that the density of each fluid element remains constant,
- Fully turbulent flow,
- Isothermal state. This consideration means that there are no temperature changes and thus the properties of the fluid are considered constant,
- The fluid used for all cases mathematically simulated, was water. This is because the water and the liquid steel its kinematic viscosity are very close,
- Gravitational forces act only on the Y axis,
- It is not considered the presence of slag and the liquid free surface was considered flat.

3.1.4 Terms of Simulation.

Later, in Fluent® the mesh code is imported, the working fluids are defined; in this case, water alone, and the input speed values of the fluid were provided to the system. The governing equations as well as the most suitable turbulence model, selected for this research were: the continuity equation, the Navier-Stokes equations, the $k - \epsilon$ turbulent flows for both groups, and the standard turbulence model. The initial values for the parameters $k - \epsilon$ are obtained from the following formulas [6]:

$$k = 0.0384 * \frac{(V_{in}^2)}{Re^{1/4}} \quad (1)$$

$$\epsilon = 2.3473 \frac{(k^{3/2})}{D_{in}} \quad (2)$$

The numerical values for these parameters are:

$$V_{in} = 0.55 \text{m/seg}; D_{in} = 15.5 \text{mm}; \kappa = 1.195 * 10^{-3}; \epsilon = 6.2564 * 10^{-3}$$

The initial values of the turbulence parameters are important to facilitate convergence in the numerical solution of the governing equations.

3.1.5 Numerical solution.

The numerical solution of the equations was carried out through the program Fluent® using the segregation method. The coupling algorithm between the pressure and velocity is called SIMPLE (Semi Implicit Method for Pressure Linked Equation). The selected convergence criteria for residuals were 10⁻³. The relaxation factors for the parameters: pressure, velocity and turbulence were 0.3, 0.7 and 0.8 respectively.

3.1.6. Mathematical simulation of residence time.

After obtaining the velocity distribution at steady state, the next step is the simulation of the distribution of residence times. This simulation is performed in unstable state being the concentration equation as follows [9]:

$$\frac{\partial(\rho C)}{\partial t} + \frac{\partial(\rho \bar{v}_i C)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{\mu}{S_{ct}} \frac{\partial C}{\partial x_i} \right) \quad (3)$$

The injection of the tracer is simulated giving a time step of 1 second and then it is simulated its diffusion through the tundish. The convergence criterion for the above equation is 10⁻⁵. The monitoring of this data is accomplished by placing a virtual sampling station in the center of each of the outputs. The monitored property is the molar concentration of the tracer. The data of the molar concentration versus time are stored in txt files, one for each of the monitored outputs.

IV. Experiments

Two capacities of a modified tundish were mathematically simulated, 40 and 60 tons. Table 1 shows an overview of the experimental conditions conducted in this research.

Table 1. Experimental conditions for the mathematical simulation of a novel tundish.

	Capacity	Mathematical Model	Geometry	Mesh (Cells)
Case 1	60 Tons	Carried out	Full	76000
Case 2	40 Tons	Carried out	Full	57000
Case 3			A quarter	15000
Case 4	40 Tons with reduction	Carried out	A quarter	31000

4.1 Results analysis.

From the mathematical simulations carried out in this research, the main task was to observe the fluid dynamic behavior of the steel in this newfangled tundish geometry.

4.1.1 60 Tons tundish.

The first stage of the research involved the study of the tundish fluid dynamic obtained directly from the modified straight type distributor. The capacity of the proposed tundish is 60 tons, which represents an increase in capacity of 50% compared with the capacity of the straight type tundish. Virtual planes, created on the tundish, once the simulation has converged, describe the fluid dynamic system. Flow patterns can be observed mainly through vectors and velocity contours. In Figure 3 can be seen that, firstly on the central longitudinal plane of the distributor and the near to the wall longitudinal plane, there is clearly visible how the fluid tends to advance mainly through the sidewalls of the distributor and not through the bottom. This is confirmed by observing how the fluid hits the bottom of the tundish and part of it tends to move down and then hits the encountering fluid oncoming. The movement of fluid into the inlet is caused by the suction of the jet. In addition, it can also display the very low speeds in the spherical cap area, which is a zone of high dead volumes.

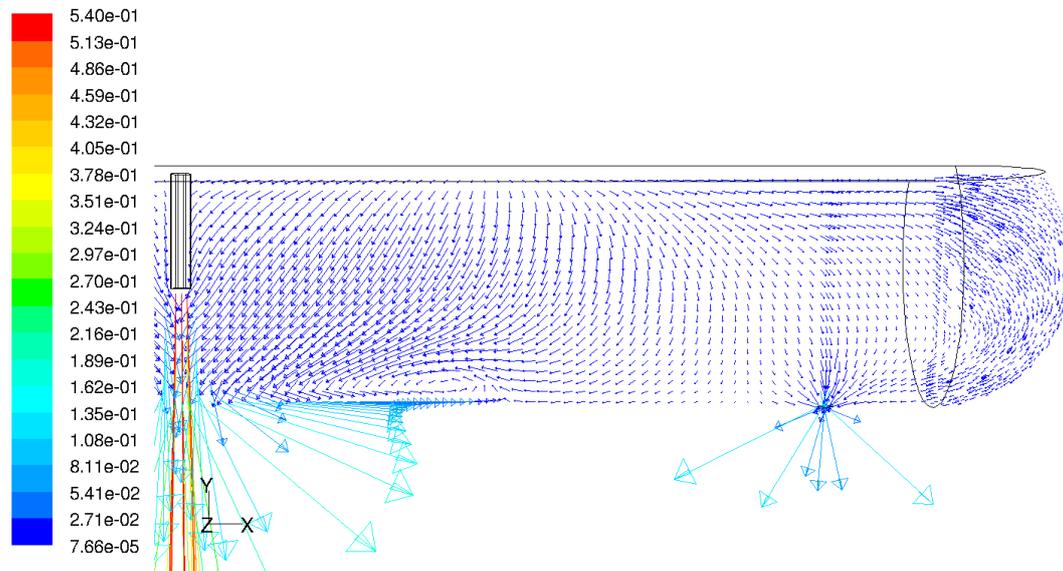


Figure 3.- Results from the mathematical simulation. Behavior of the fluid flow along the central longitudinal plane of the tundish.

In the zone corresponding to the free region of the tundish (semi-spherical area), the fluid has an asymmetric behavior resulting in recirculating flows as reported also by Torres-Alonso and co-workers⁽⁹⁾. Figure 4 shows a well defined recirculation at one side, and on the opposite side there is seen a certain recirculation that is not well defined, this is attributed more to an unstructured meshing than the physical phenomenon itself; besides to this, it also confirms that this is an area of very low speed. The unstructured mesh in this area of the tundish implies that if a line of symmetry is drawn in the spherical cap, each side of it has a different number of mesh cells. Recall that the application of a structured mesh in this area is not feasible due to the complex geometry of this part of the tundish.

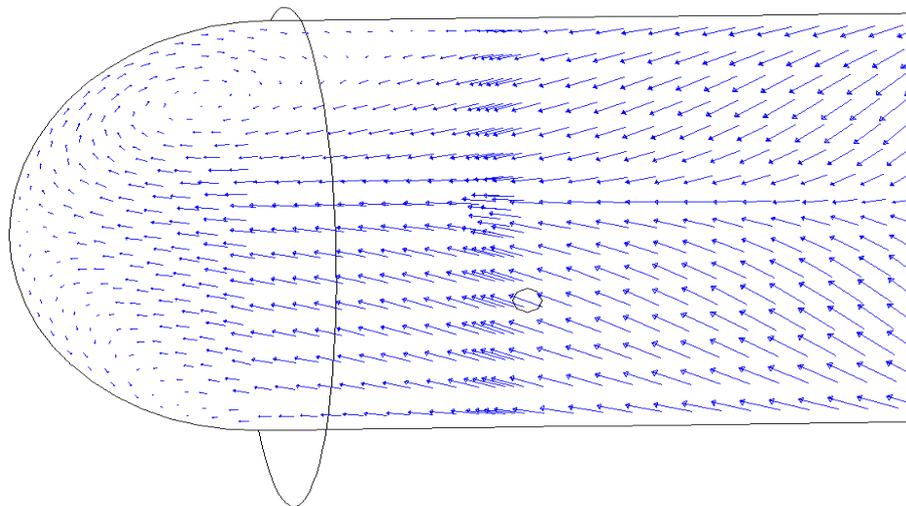


Figure 4.- Asymmetric behavior of the fluid on the free surface, With recirculation at the top of the hemispherical cap and non asymmetric flow at the bottom

After obtaining the patterns of fluid flow in the distributor, the next step is to characterize this flow. The characterization of the fluid flow by means of the mathematical simulation is obtained by monitoring (*virtually*) the concentration of a tracer, which is injected at the input current, in each of the outputs of the distributor. Fractions of the flow for each of the outputs are shown in Table II.

Table II. Mathematical characterization of fluid flow for the 60 tons tundish

	Piston volume	Mixed volume	Dead volume
Right exit	34.8 %	42.6 %	22.6 %
Left exit	33.4%	43.9%	22.7%

4.1.2 40 Tons tundish.

For this tundish capacity as well as a full quarter of it was simulated, finding the same asymmetric behavior in the area of the hemispherical caps for the full geometry, which can be seen in Figure 5, in which use of a special software function is made, which assigns the same size for all the vectors, for a better way to visualize the aforementioned behavior. This prevailing asymmetrical condition was the main reason to simulate the system using the symmetry condition.

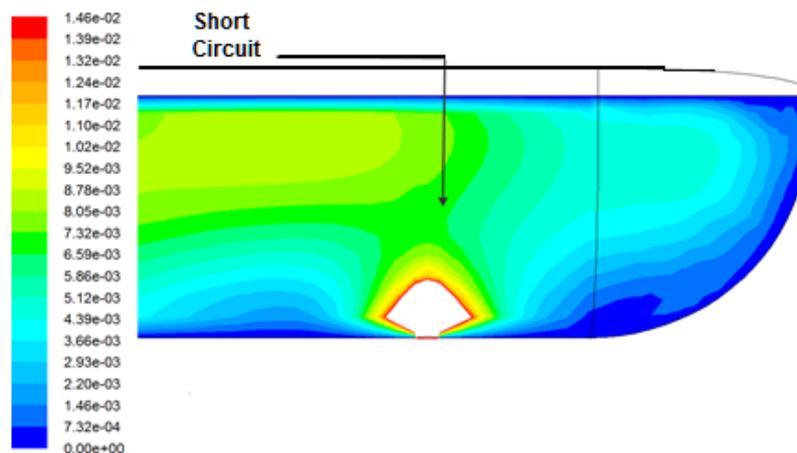


Figure 5.- Velocity contours in the central longitudinal plane (40 tons).

The overall performance of the fluid is the same as for the 60 tons capacity; however, for this capacity the fluid shows a higher fluid dynamic due to a decrease in the level of steel in the tundish. The presence of areas of greater speed in the 40 tons tundish is caused by the fact that the fluid has a smaller contact area with the sidewalls of the tundish in its ascent to the surface, losing less speed. On the other hand, downgrading causes a short circuit, which is a common phenomenon when decreases the level of the liquid steel.

For the characterization of the flow, for the 40 tons tundish, it was virtually monitored only one output, due to the symmetry condition used. Fractions of the fluid flow for this tundish capacity are shown in Table III.

Table III. Results from the mathematical characterization of the fluid flow.

Capacity (Ton.)	Piston Volume	Mixed Volume	Dead Volume
40	39.5 %	42.4 %	18.1%
60	34.1%	43.2%	22.6%

The fraction of dead volume for this capacity is practically the same as that reported by Barreto et al [2] for a straight type tundish, which is a high fraction of dead volume. By decreasing the capacity of the tundish also decreases the dead volume fraction at about 20%.

4.1.3 Tundish longitudinal reduction.

Due to the high death volume in the system and because it is virtually the same fraction as the one reported for straight type tundish, and which does not represent a significant improvement, it was proposed to reduce longitudinally the model, as shown in Figure 6. Once this part of the model is reduced it causes in the area of semi-spherical caps a greater dynamic, because the fluid arrives with higher energy in that area, and therefore the dead volume is reduced. The reduction in the dead volume fraction was reduced by approximately 11% compared to the value in the tundish without reduction. The values of the flow fractions are presented in Table IV.

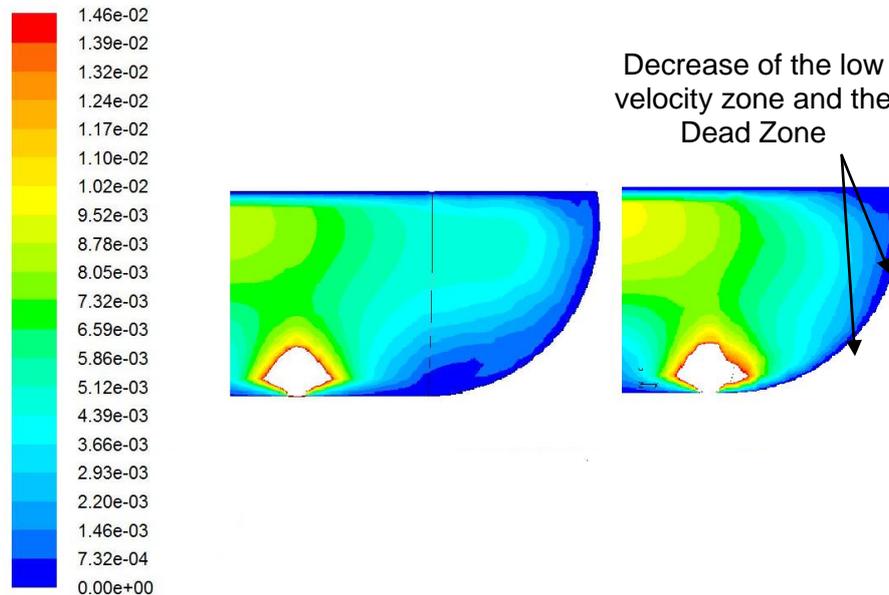


Figure 6.- Comparison of speed contours between the reduced tundish and the one without reduction.

Table IV. Dead volume results according to the fluid flow regime in the tundish.

	Piston Volume	Mixed Volume	Dead Volume
Mathematical model	37.5%	46.4%	16.1%

4.1.4 Comparison with the straight type tundish.

Comparison among the modified tundish with the straight type was mainly focused on two aspects: the influence of geometry on the stability of the entrance area and the characterization of the fluid flow. The stability of the area plays an important role in the design of a tundish since it affects the reduction of refractory wear and avoid opening the slag layer due to high turbulence, helping to get steel more clean and homogeneous in both composition and temperature.

Figures 7 and 8 show the difference of the impact of jet entry into the rectum distributor type [2] and the proposed one, in the bottom and sidewalls respectively.

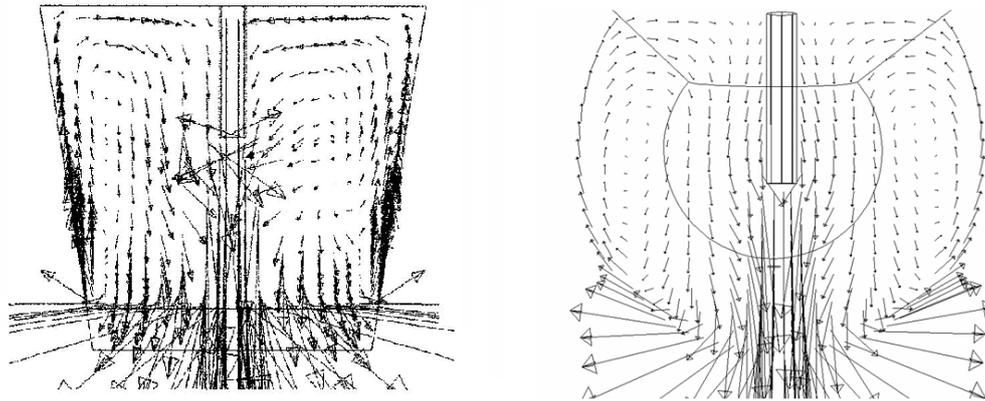


Figure 7.- Comparison of the entry jet impact at the bottom of the straight type tundish (right) ^[6] and the (left) proposed one.

In the straight type tundish occurs a violent impact with the bottom of it because it has a flat base, this impact causes the steel to spread in all directions; also the impact of the fluid against the sidewalls of the straight type tundish is also very strong. On the other hand in the proposed tundish the impact of the jet inlet is moderated due to the cylindrical geometry of the tundish. This also bring on that the fluid gradually ascends the sidewalls, reducing the impact with them.

Decreasing the speed at which the steel hits the walls of the tundish could reduce refractory wear and therefore reduce the amount of non-metallic inclusions, resulting in cleaner steel.

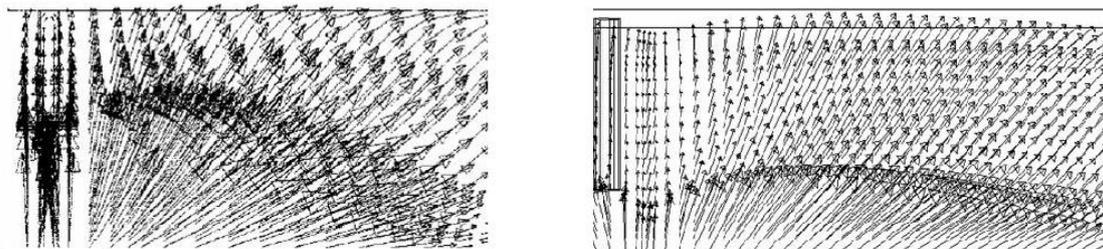


Figure 8.- Impact on the sidewalls of the straight type tundish (right) ^[2] and the proposed one (left).

4.1.5 Quantitative comparison.

The dead volume fraction is the most critical parameter in the characterization of the fluid flow and thus is the primary indication of system efficiency. In Table V a comparison of flow fractions is presented, mathematically obtained, between the two-tundish designs, the straight type studied by Barreto et al, ^[2] and the proposed one in this research.

Table V. Comparison of the flow fractions between the two-tundish designs.

Tundish type	% Dead Volume	% Piston Volume	% Mixed Volume	$\frac{V_{pv}}{V_{mv}}$
Straight ^[2]	18.8	20.1	61	0.33
Proposed	18.1	39.5	42.4	0.93
Proposed: longitudinally reduced	16.1	37.5	46.4	0.81
Straight with turbulence inhibitor. ^[2]	11.1	35.60	53.10	0.67

As can be seen in the table, the dead volume fraction between the straight type tundish and the proposed one without reduction, both with a capacity of 40 tons, is practically the same, varying in 3.7%, it was the main reason for reducing the tundish, and seeks for a lower dead volume fraction. However with the reduction in size of the proposed tundish this fraction decreases only 15% with respect to that of the straight type tundish.

Another important aspect is that in the configuration of the proposed tundish *no use of an inhibitor of turbulence* is made, because the cylindrical shape of the bottom of the tundish performs this function, so comparing the proposed tundish, reduced in size, with the straight type with turbulence inhibitors, it can be observed that the fraction of dead volume is 31% higher which represents a very significant difference, however having no turbulence inhibitors may significantly reduce maintenance costs and also reducing the possibility of

generation of inclusions. Now it could be introduced a turbulence inhibitor and improve even more the performance of this new tundish design, but without doubt, this would already be the subject of another research project.

In contrast an extremely favorable parameter in the proposed tundish is the high relationship between the piston and mixed flow fractions, with respect to the relationship existed in the straight type distributor. This parameter is a clear indication of how stable the entrance area of the tundish is, a very important feature in the removal efficiency of inclusions is concerned.

VI. Conclusions

1. The concave geometry of the bottom of the tundish decreases the turbulence in the discharge area of the liquid steel, reducing the magnitude of the input stream impact with the walls of the tundish. This causes the fluid to rise through the walls of a contained manner, helping to reduce erosion of the refractory in this area, and make it more stable, thereby preventing the opening of the slag layer.
2. The hemispherical ends of the tundish are zones with very low speeds of fluid flow, which are regions that have a large part of the dead volume present in the system. However, in the modified tundish, reduced in size, this phenomenon is reduced so with an improved tundish design it could better exploit the beneficial effect of the hemispherical walls, since one of the benefits presented by this design is that it reduces the impact with the sidewalls of the tundish which eliminates the phenomenon of vorticity. With this it could then be considered increasing the capacity of the tundish, i.e., increase the tonnage of steel to cast.
3. The fraction of dead volume in the proposed tundish (without reduction) is practically the same as that reported for the straight type tundish. However, reducing longitudinally the tundish, the dead volume fraction is also reduced in about 15%. However, the piston volume fraction is greater, compared to straight type tundish, which is clear indication of the inhibitory effect of the concave shape of the bottom of the tundish.
4. Increasing the capacity of the tundish does not imply greater productivity of the system, once this increase also brings an increase in the fraction of dead volume; nevertheless, with the introduction into the tundish of a "turbulence modifier", the flow would be redirected towards the ends, causing a larger dynamic in this area and thereby reduce the percentage of dead volume.
5. The use of an unstructured mesh in the hemispherical caps zone causes the appearance of asymmetric flows in said area. The modeling of a section of the same, only a quarter of the model is an excellent choice to circumvent this problem.
6. Implementing geometric cuts on the system reduces the deformation of the cells and contributes to better outcomes.

Bibliography

- [1]. Irving. W.R. "Continuous Casting of Steel", Editorial Cambridge, (1993), pp. 1-38.
- [2]. Lopez-Ramirez S., Barreto J. de J., Palafox-Ramos J., Morales R.D. y Zacharias D. "Modeling Study of the Study of Turbulence Inhibitors on the Molten Steel Flow, Tracer Dispersion and Inclusion Trajectories in Tundishes". Metallurgical and Materials Transactions B Volume 32b, August 2001.
- [3]. Barrera G. y Barreto J. de J. "Análisis del desempeño Hidrodinámico Anisotérmico del Distribuidor de la Máquina de Colada Continua del acero.". Revista IDEAS, UAEM, (1997) pp. 2-7.
- [4]. Jha P K., Dash K. S. y Kumar S. "Fluid Flow and Mixing in a Six Strand Billet Caster Tundish: A parametric Study" ISIJ International, Vol. 41 (2001), No. 12, pp. 1437-1446.
- [5]. Mazumdar D. y Guthrie R. "The Physical and Mathematical Modelling of Continuous Casting Tundish Systems." ISIJ International, Vol. 39 (1999), No. 6, pp. 524-567.
- [6]. Tripathi A. y Ajmani. S. K. "Numerical Investigation of Fluid Flow Phenomenon in a Curved in a Curved Shape Tundish of Billet Caster" ISIJ International, Vol. 45 (2005), No. 11, pp. 1616-1625.
- [7]. Tripathi A. y Ajmani. S. K. Effect of shape and flow control devices on the fluid flow characteristics in three different industrial six strand billet caster tundish. ISIJ International (2011), Vol. 51, No. 10, pp 1647-1656.
- [8]. Ilegbusi O. Iguchi M. y Wahnsiedler W. "Mathematical and Physical Modelling of Materials Processing Operations" Ed. Chapman & Hall (2000) pp. 2-26.
- [9]. Torres-Alonso E., Morales R., García-Hernández S., Nájera-bastida A., Sandoval-Ramos A. Influence of straight nozzles on fluid flow in mold and billet quality. Metallurgical and Materials Transactions B. (2008), Vol.39, No. 6, pp 840-852.