Modeling of Fluid Flow and Residence Time Distribution in a Four-strand Tundish for Enhancing Inclusion Removal

Anil KUMAR,1) Dipak MAZUMDAR2) and Satish C. KORIA2)

1) Steelmaking and Casting Research Group, Research and Development Division, Tata Steel, Jamshedpur–831 007, India. E-mail: dr.anilkumar@tatasteel.com 2) Department of Materials and Metallurgical Engineering, Indian Institute of Technology, Kanpur–208 016, India. E-mail: dipak@iitk.ac.in, satishch@iitk.ac.in

(Received on August 24, 2007; accepted on November 13, 2007)

The introduction of the appropriate size and precise location of flow control devices such as dam, turbulence inhibitor, etc., helps to modify the flow pattern and minimizing short circuiting and dead zone. Beside this, these also create the surface directed flow and maximize the residence time available for the flotation of inclusions and assimilation of the reaction products from the molten steel into the slag phase. These can be products of deoxidation, reoxidation, precipitation, emulsification and/or entrainment of refractory components into the melt and thus encompass both indigenous and exogenous inclusions. To this end, both the numerical and physical simulations were carried out mainly for three cases: a) in absence of flow control devices (i.e., bare tundish), b) in the presence of a dam, and c) with the application of turbulence inhibiting device (TID) and dam combination in the existing tundish configuration. The commercial CFD (computational fluid dynamics) package FLUENT® was used to predict the flow field prevalent in the water model tundish at steady state, whereas in the experimental program, both Particle Image Velocimetry (PIV) techniques for flow measurements and tracer dispersion experiments for concentration measurements were applied in the present study. Among all types of configurations applied in the present study, a combination of TID with holes+a dam work reasonably found to be an optimum configuration of the four-strand tundish regarding inclusion floatation. A superior strand similarity is also achieved in this configuration. Also the predicted time averaged horizontal and vertical components agreed within ±10% with the experimentally derived ones.

KEY WORDS: fluid flow; flow pattern; flow control devices; short-circuiting; residence time distribution; particle image velocimetry; tracer dispersion experiments; physical and mathematical modeling; horizontal and vertical velocity components; strand similarity.

1. Introduction

In the initial stages of development of the tundish, it acted as a reservoir1,8) to hold sufficient liquid steel so as to provide a constant head over the mold and to permit a ladle exchange to occur without interruption of sequence casting. Today, the continuous casting tundish has evolved into a useful reactor for liquid steel refining. As such, it now has important roles to play over and above its traditional role as a buffer or steel distribution vessel. Thus a modern day steelmaking tundish is designed to provide maximum opportunity for carrying out various metallurgical operations such as inclusion removal, alloy trimming of steel, calcium doped inclusion modification, and superheat control, thermal and particulate homogenization, etc. The optimum tundish configuration along with an optimum ladle shroud depth prevents the entrapment of slag droplets through inlet jet into the system. Elimination of contamination by air and/or refractory phases and technologies to support clean steel production is part and parcel of sound tundish operations. On the other hand, alloy dissolution and distribution of the dissolved product is required intense mixing in the tundish bath. The state of fluid motion in tundish considerably influences the rate of such heat and mass transfer controlled processes. As majority of metallurgical operations are governed by heat and mass transfer, consequently, the nature of the fluid flow (viz., spatial velocity distribution, turbulent kinetic energy etc.) influences tundish performance considerably.

Inclusions in tundishes generally have their source in the carry over slag from the ladle (in the form of microdroplets), in the entrained of tundish slag, eroded refractory wall, chemical/steel deoxidation reactions etc. Many studies have shown that the basic tundish design, operating parameters (viz., throughput rate, bath depth, inflow rates, etc.), as well as various flow modifiers influence residence time distribution (RTD) values in tundishes and therefore, exert considerable influence on the efficiency of inclusion separation as well as its floatation. It is now rather well known that a tundish without any flow modifier do not satisfy the requirements essential to inclusion removal and consequently, critical to the production of clean steel. The bare tundishes are associated with considerable short circuiting,1–8) large dead volumes3,9) significant turbulence,10) and
slag droplet entrainment, etc., which are potentially detri-
mental to the floatation of non metallic inclusions. It must
however, be mentioned that specific conditions for removal
of inclusions vary largely from one practice to another, the
size of inclusions being a critical factor in determining the
effectiveness of a given tundish design to separate and float
out inclusions. It is suggested\textsuperscript{11} that inclusions with a large
terminal velocity or larger particle diameter will readily
float-up to the slag-metal interface provided appropriate
flow modifiers are incorporated into the tundish geometry.
As the existing or basic tundish geometry can not be fre-
quently interchanged/revamped at the caster (due to ma-
chine constraints or so) to fulfill the steelmaker’s require-
ments accordingly, and therefore, the insertion of the appro-
priate flow control devices in the tundish is the best option
to create desirable flow pattern economically. Numerous
studies have been reported in the literature on fluid flow and
RTD in steelmaking tundish. A majority of these studies
was, however, restricted to a single or two-strand slab
caster. Thus, not much information on multi-strand tundish
regarding fluid flow and RTD is available in the literature
to understand the strand variations during the continuous cast-
ing and tundish performance along with inclusion removal.
Therefore, the present study aims to investigate the tundish
configuration leading to promote the inclusion floatation
with minimizing dead regions as well as strand variations
of the four-strand tundish system.

2. Mathematical Modeling

The flow produced in a four-strand steelmaking tundish
by a submerged liquid steel stream is simulated through
water flowing in a Perspex glass tundish. Figure 1 schemat-
ically shows the model tundish with various physical
boundaries. The dimensions and the operating parameters
of the tundish, deduced via geometrical and dynamic simi-
larity conditions, are summarized in Table 1 along with
those of the full-scale system. The full-scale tundish is
filled with a turbulence inhibiting device (TID) and a dam.
Accordingly, the model tundish was fabricated to have
complete geometrical similarity with the full-scale system.
The design of TID for the model tundish is shown in Fig. 2.
The TID was manufactured from 10 mm thick glass sheet.
As seen, it has 170 mm long, 210 mm wide and 52 mm high
with 20 mm top flange and 10 mm×30 mm rectangular side
holes. The purpose of the holes of TID was to accelerate
the surface directed flow beside TID and far away from the
inlet region and thus minimizing the dead zones as well as
temperature distributions in the tundish. It is placed at the
centre of the tundish base and just beneath the inlet stream
dot.

2.1. Model Formulation

The commercial CFD package FLUENT\textsuperscript{\textregistered} was used to
predict the flow field prevalent in the water model tundish
at steady state. The following assumptions were made:
\begin{itemize}
\item[a)] Cartesian coordinate system in three dimensions has
been used to represent the tundish geometry and the
governing equations.
\item[b)] Flow has been assumed to be dynamically steady and
three-dimensional.
\end{itemize}

\begin{table}[h]
\centering
\begin{tabular}{|l|l|l|}
\hline
Tundish parameters & Industrial tundish & Model tundish \\
\hline
Length, m & 3.0 – 8.0 & 1.0 \\
Width×Length & 0.10 – 0.30 & 0.23 \\
Bath height×Length & 0.10 – 0.30 & 0.26 \\
Exit distance×Length from inlet & 0.38 – 0.92 & 0.49 – 0.88 \\
Depth of submerged shroud×Bath height & 0.08 – 0.62 & 0.15 – 0.45 \\
Side walls inclination & 0° – 15° & 10° \\
No. of strands & 2 – 6 & 4 \\
Volumetric flow rate, m\textsuperscript{3}/s & 1.18×10\textsuperscript{-3} – 2.8×10\textsuperscript{-4} & 1.55×10\textsuperscript{-4} \\
Froude number & 0.45 – 111 & 1.24 \\
Reynolds number & 10\textsuperscript{3} – 10\textsuperscript{5} & 10\textsuperscript{5} \\
\hline
\end{tabular}
\caption{Characteristic parameters of industrial\textsuperscript{12} and model
tundish.}
\end{table}

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig1.png}
\caption{A schematic of the model tundish showing the various
physical boundaries.}
\end{figure}

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig2.png}
\caption{Dimensions of TID with rectangular top flange with side
holes.}
\end{figure}

\begin{itemize}
\item[c)] Flow is turbulent through out the tundish.
\item[d)] The tundish is operated under isothermal conditions. It
has been observed that the temperature distribution in
the tundish is depending upon the tundish configura-
tion and varied upto 5°C, which can be reduced signifi-
cantly by minimizing the dead regions and protecting
the heat losses in the tundish. However, some extent of
temperature distribution is favorable for inclusion re-
moval.\textsuperscript{13} In this context, it is expected that the tem-
perature distribution of real tundish for optimal configura-
tion is marginal and is, therefore, assumed very close

© 2008 ISIJ
to the isothermal condition.

e) Thermophysical properties of the fluid are taken to be constant.

The associated governing equations (continuity, momentum, equations for turbulence and tracer dispersion equation) can be written in Cartesian coordinate system (x,y,z) under steady state condition as:

2.1.1. Equation of Continuity

\[ \frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) + \frac{\partial}{\partial z} (\rho w) = 0 \quad \text{(1)} \]

2.1.2. Equations of Momentum

\begin{align}
\text{x-momentum} \\
\frac{\partial}{\partial x} (\rho u u) + \frac{\partial}{\partial y} (\rho u v) + \frac{\partial}{\partial z} (\rho u w) \\
&= -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( \mu_\text{eff} \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_\text{eff} \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_\text{eff} \frac{\partial u}{\partial z} \right) \\
&\quad + \frac{\partial}{\partial x} \left( \mu_\text{T} \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_\text{T} \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_\text{T} \frac{\partial u}{\partial z} \right) \\
\text{y-momentum} \\
\frac{\partial}{\partial x} (\rho u v) + \frac{\partial}{\partial y} (\rho v v) + \frac{\partial}{\partial z} (\rho v w) \\
&= -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left( \mu_\text{eff} \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_\text{eff} \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_\text{eff} \frac{\partial v}{\partial z} \right) \\
&\quad + \frac{\partial}{\partial x} \left( \mu_\text{T} \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_\text{T} \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_\text{T} \frac{\partial v}{\partial z} \right) \\
\text{z-momentum} \\
\frac{\partial}{\partial x} (\rho u w) + \frac{\partial}{\partial y} (\rho w v) + \frac{\partial}{\partial z} (\rho w w) \\
&= -\frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left( \mu_\text{eff} \frac{\partial w}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_\text{eff} \frac{\partial w}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_\text{eff} \frac{\partial w}{\partial z} \right) \\
&\quad + \frac{\partial}{\partial x} \left( \mu_\text{T} \frac{\partial w}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_\text{T} \frac{\partial w}{\partial y} \right) + \frac{\partial}{\partial z} \left( \mu_\text{T} \frac{\partial w}{\partial z} \right) \\
\end{align}

Equations (2) to (4) are called Reynolds-averaged Navier–Stokes (RANS) equations. The equations contain additional unknown variables, \( \mu_\text{T} \) (turbulent viscosity) and hence \( \mu_\text{eff} \) (effective viscosity = \( \mu_\text{T} + \mu \)) and therefore, turbulence models are needed to determine these variables in terms of relevant quantities.

2.1.3. Standard Coefficient \( k-\varepsilon \) Model

Launder and Spalding\(^6\) proposed the standard coefficient \( k-\varepsilon \) model. It comprises of two separate transport equations describing the conservation of both turbulent velocity and length scales. Robustness and reasonable accuracy of the model to a wide range of turbulent flows explain its popularity in the simulation industrial flow and heat transfer. The standard coefficient \( k-\varepsilon \) model is a semi-empirical model based on a modeled transport equations for the isotropic turbulence kinetic energy \((k=3/2(\varepsilon^2))\) and its dissipation rate \((\varepsilon=-d\varepsilon/dt)\). The turbulence kinetic energy and its rate of dissipation are obtained from the following transport equations:

- Conservation equations for the turbulence kinetic energy:

\[ \frac{\partial}{\partial x} (\rho u k) + \frac{\partial}{\partial y} (\rho v k) + \frac{\partial}{\partial z} (\rho w k) \]

\[ = \frac{\partial}{\partial x} \left( \frac{\mu_\text{eff}}{\sigma_k} \frac{\partial k}{\partial x} \right) + \frac{\partial}{\partial y} \left( \frac{\mu_\text{eff}}{\sigma_k} \frac{\partial k}{\partial y} \right) + \frac{\partial}{\partial z} \left( \frac{\mu_\text{eff}}{\sigma_k} \frac{\partial k}{\partial z} \right) \]

\[ + G_k - \rho \varepsilon \quad \text{...........................................(5)} \]

- Conservation equations for the dissipation rate of turbulence kinetic energy:

\[ \frac{\partial}{\partial x} (\rho u \varepsilon) + \frac{\partial}{\partial y} (\rho v \varepsilon) + \frac{\partial}{\partial z} (\rho w \varepsilon) \]

\[ = \frac{\partial}{\partial x} \left( \frac{\mu_\text{eff}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x} \right) + \frac{\partial}{\partial y} \left( \frac{\mu_\text{eff}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial y} \right) + \frac{\partial}{\partial z} \left( \frac{\mu_\text{eff}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial z} \right) \]

\[ + \frac{1}{k} (C_1 k \varepsilon - C_2 \rho \varepsilon^2) \quad \text{...........................................(6)} \]

In these equations, \( G_k \) represents the generation of turbulence kinetic energy due to the mean velocity gradients and can be expressed as:

\[ G_k = 2 \mu_\text{T} \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right] \]

\[ + \mu_\text{T} \left[ \left( \frac{\partial \varepsilon}{\partial x} \right)^2 + \left( \frac{\partial \varepsilon}{\partial y} \right)^2 + \left( \frac{\partial \varepsilon}{\partial z} \right)^2 \right] \]

\[ \quad \text{...........................................(7)} \]

The turbulent (or eddy) viscosity, \( \mu_\text{T} \), is computed by combining \( k \) and \( \varepsilon \) as follows:

\[ \mu_\text{T} = \frac{C_\mu \rho k^2}{\varepsilon} \quad \text{...........................................(8)} \]

\( C_1, C_2, C_\mu, \sigma_k, \) and \( \sigma_\varepsilon \) are the five empirical constants of the \( k-\varepsilon \) model. The standard values\(^6\) of these constants are 1.43, 1.92, 0.09, 1.0 and 1.3 respectively.

2.1.4. Tracer Dispersion Equation

To evaluate the process performance of continuous cast-
ing tundish systems mathematically, RTD has been predicted under a wide variety of conditions by simulating a pulse injection of an inert tracer into the inlet stream of a tundish and the variation of concentration of an injected tracer at the exit of the tundish was estimated as a function of time by solving a transient, three dimensional, convection + diffusion equation. In the presence of three dimensional velocity field $u$, $v$, and $w$ with no generation, the conservation of an added tracer concentration, $C$ (kg/m$^3$), expressed in terms of the Cartesian coordinates, according to:

$$\frac{\partial C}{\partial t} + \frac{\partial (uC)}{\partial x} + \frac{\partial (vC)}{\partial y} + \frac{\partial (wC)}{\partial z} = \frac{\partial}{\partial x} \left( D_{eff} \frac{\partial C}{\partial x} \right) + \frac{\partial}{\partial y} \left( D_{eff} \frac{\partial C}{\partial y} \right) + \frac{\partial}{\partial z} \left( D_{eff} \frac{\partial C}{\partial z} \right)$$

...............................................(9)

In this equation, $D_{eff}$ the effective diffusivity is the sum of the molecular and eddy diffusivity ($D + D_T$). The eddy diffusivity is taken to be numerically equal to the eddy kinematic viscosity ($=\mu/\rho$), assuming the turbulent Schmidt number ($=\mu_{eff}/\rho D_{eff}$) to be unity.

2.2. Boundary Conditions

Following boundary conditions were used:

1. The formation of waves at the free surface was ignored. The free surface was assumed to be flat and mobile. Fluxes of all quantities across the free surface were assumed to be zero. Therefore, normal velocity component (for convective flux) and normal gradients of all variables (for diffusive flux) were all set to zero, i.e.,

$$w = 0, \quad \frac{\partial u}{\partial z} = 0, \quad \frac{\partial w}{\partial z} = 0, \quad \frac{\partial k}{\partial z} = 0, \quad \frac{\partial \varepsilon}{\partial z} = 0,$$

and

$$\frac{\partial C}{\partial z} = 0 \quad \text{.............}(10)$$

2. The tundish exit can be computationally treated as either a standard outflow or as a plane or surface, at which flow occurs at an ambient pressure (taken). At the tundish outlets, both types of boundary conditions were applied in order to assess the similarity of the experimental results to model configuration.

3. At all the solid walls, the velocity components and flux of tracer were all set to zero, viz., at both the side walls,

$$u = 0, \quad v = 0, \quad w = 0, \quad k = 0, \quad \varepsilon = 0, \quad \text{and} \quad \frac{\partial C}{\partial x} = 0$$

...............................................(11)

at both the frontal side walls,

$$u = 0, \quad v = 0, \quad w = 0, \quad k = 0, \quad \varepsilon = 0, \quad \text{and} \quad \frac{\partial C}{\partial y} = 0$$

...............................................(12)

at the bottom wall

$$u = 0, \quad v = 0, \quad w = 0, \quad k = 0, \quad \varepsilon = 0, \quad \text{and} \quad \frac{\partial C}{\partial z} = 0$$

...............................................(13)

For the simulation of flow in the near wall region universal logarithmic law was applied on the parallel to wall flow components:

$$\frac{p_{out} k^{1/2} C_{1/4}}{\tau_w} = \frac{1}{\kappa} \ln(y^+) \quad \text{.................}(14)$$

where,

$$y^+ = y/v$$

...............................................(15)

In the above expressions, $E$ is the empirical constant (=9.79); $\kappa$ is the Von Karman's constant (=0.42); $\tau_w$ is the shear stress on the wall; $u_e$ is the resultant velocity of the fluid near the wall; $y^+$ is the dimensionless normal distance from the resultant velocity; $y_p$ is the distance of the first node point $p$ from the wall. Similar equations were utilized for the first grid points adjacent to the frontal side walls (longitudinal) and side walls (transverse), with the term $y_p$ being replaced by $z_p$ and $z_{pp}$ respectively.

4. At the inlet plane, normal velocity (deduced on the basis of the mass inflow rate and shroud dimension) and turbulence parameters were prescribed. Accordingly, $k=0.01 U_{in}^2$ and $\varepsilon=2k^{3/2}/d$ were specified at the flow inlet. In most of the computational studies, the location of the inlet boundary was chosen at the port/meniscus region. Few authors have applied ladle outlet also. It is apparent that prescription of a flat velocity profile at the port/meniscus region is not a suitable representation of the actual reality as can be seen in the earlier publication. The frictional forces acting on the flowing fluid by the walls of the nozzle is likely to alter the velocity profile; thus ladle outlet appears to be a realistic location of the inlet boundary (see Fig. 1).

2.3. Numerical Solution Procedure

A non uniform grid system with varying grid densities as applied in the inlet and outlets as well as the bulk of the tundish. On an average, altogether, 150 000 cells were used to encompass the calculation domain (1.0 m×0.5 m×0.3 m). An implicit scheme with segregated solver was applied. In the numerical solution scheme Semi Implicit Method for Pressure Linked Equation (SIMPLE) algorithm was used for pressure velocity coupling and first order upwind scheme for momentum and scalar transport equations. The convergence criterions for scaled residuals were set to be less than $10^{-3}$ except for concentration which is set to $\leq 10^{-5}$. Under relaxation factors for pressure ($\alpha_p$), momentum ($\alpha_{momentum}$) and specific kinetic energy and its dissipation rate ($\alpha_{\varepsilon}$) were applied as 0.3, 0.7 and 0.8 respectively, to seek the numerical solution. Once the flow field was converged to steady state the problem defined module of the FLUENT solver was changed to unsteady state for solving the transient tracer dispersion Eq. (9) with appropriate initial and boundary conditions. Consequently,
at $t=0$, a small volume element was assumed to be filled with the inert tracer at the inlet plane as a pulse and subsequently, at $t>0$, the tracer concentration was predicted computationally for various instants of time at the outlet plane of the tundish to obtain the C curve.

3. Physical Modeling

In the experimental program, both Particle Image Velocimetry (PIV) techniques for flow measurements and tracer dispersion experiments for concentration measurements were applied in the present study. The physical properties for water at 20°C and liquid steel at operating temperature (∼1 600°C) are listed in Table 2.

### 3.1. Flow Measurement

Figure 3 shows a schematic of the complete experimental set-up used for velocity measurement. As shown, it comprises a model tundish, water supply system, laser beam, PIV instrumentation, CCD camera and a mini computer with 10 MB RAM. The manufacturer of this PIV instrument is Tesscorn System Incorporated. In the beginning, the tundish is operated under steady state conditions with negligible fluctuation at the free surface. Subsequently the flowing water within the tundish is seeded with hollow glass spheres (<50 μm size and density of 1 030 kg m$^{-3}$) to trace the path of the flowing fluid. The terminal-rise velocity of the 50 μm glass sphere particle is calculated to be 0.04 mm/s. The flow is then illuminated in the target area (essentially a two dimensional slice) with a laser light sheet.

![Diagram](image)

**Fig. 3.** A schematic of PIV setup for two-dimensional velocity measurements in water flowing tundish system.

### Table 2. Physical properties of fluid used in the present investigation and their comparison with those in actual system.

<table>
<thead>
<tr>
<th>Physical properties</th>
<th>Liquid steel at 1600°C</th>
<th>Water at 20°C</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$, kgm$^{-3}$</td>
<td>7000</td>
<td>1000</td>
</tr>
<tr>
<td>$\mu$, kgm$^{-1}$s$^{-1}$</td>
<td>6.7×10$^{-4}$</td>
<td>1.1×10$^{-3}$</td>
</tr>
<tr>
<td>$\nu$, m$^{2}$/s</td>
<td>9.01×10$^{-6}$</td>
<td>1×10$^{-5}$</td>
</tr>
<tr>
<td>$\sigma$, Nm$^{-1}$</td>
<td>1.6</td>
<td>0.073</td>
</tr>
</tbody>
</table>

$\rho$ density, $\mu$ viscosity; $\nu$ kinematic viscosity; $\sigma$ surface tension

The camera lens images the target area onto the coupled charged device (CCD) array of a digital camera. The CCD is able to capture the movement of particles between two light pulses in separate image frames. Recording both light pulses in the same image frame to track the movements of the particles gives a clear visual sense of the flow pattern. Once a sequence of two light pulses is recorded, the images are divided into small subsections called interrogation areas. The interrogation areas from each image frame are cross-correlated with each other, pixel by pixel by employing the Fast Fourier Transformations (FFT) technique. The correlation function produces a signal peak and supplies values of the flow for each pixel interrogation area. The position of the highest peak of the cross correlation function is later transformed with an inverse FFT relative to the origin of the interrogation area, which corresponds to the particle displacement, $\Delta y$. On the basis of an accurate measurement of the displacement and exposure time, $\Delta y_{\text{PIV}}$, i.e., time difference between two consecutive images, the velocity vector is then achieved by dividing the measured displacement with the exposure time delay with the help of sub-pixel interpolation procedure. The local velocity vector map over the whole target area is obtained by repeating the cross-correlation for each interrogation area over the two image frames captured by the CCD camera. In a typical experiment, the CCD camera records twenty-five instantaneous images (flow field) in 0.25 s from which the time averaged representative flow field of the target area was obtained. For a given experimental condition, we repeated experiments at least twice and found practically identical results.

### 3.2. Tracer Dispersion Measurement

In addition to the PIV experiments, the tracer dispersion experiments were also performed to understand the flow pattern in the four-strand tundish. The tundish was filled with water to the depth 0.26 m. After attaining steady state, a total of 20 mL potassium chloride as a tracer (concentration 200 g/L) was injected as a pulse into the inlet stream. The conductivity of water at the exits was recorded continuously on the PC for 20 min duration at a rate of 5 Samples/s with the help of data acquisition card through individual channel. These conductivity values of individual strands were converted into corresponding concentrations and the dimensionless concentrations were plotted against dimensionless time to derive the characteristic C curves. The individual C curves effect the strand to strand variations whereas the overall C curve (average of individual C curves) is used to determine the flow volumes in multi-strand tundish system. In each experiment, mass balance of tracer was made and output of the tracer lie in between 90–96% of the input. The experimental apparatus used for the monitoring of concentration following a pulse tracer addition has been described in detail in an earlier publication and, consequently is not reiterated here.

4. Results and Discussion

Both the numerical and physical simulations were carried out mainly for three cases: a) in absence of flow control devices (i.e., bare tundish), b) in the presence of dam, and c)
with the application of TID-dam combination in the existing tundish configuration. It can be noted that $L$ and $W$, respectively, were defined as the ‘half length’ and the ‘half width’ whereas $H$ was defined as the ‘bath height’ of the tundish in the present study, and were coordinated along $X$, $Y$ and $Z$ directions respectively. The origin of the domain was kept at the centre of the base of tundish as already indicated in Fig. 1. For the second configuration, the dam height of 0.6$H$ was placed at $X/L=0.39$ equidistant both sides from the inlet whereas the dam of 0.2$H$ combination with TID (placed at the centre of the tundish base) was inserted at the position of $X/L=\pm0.39$ from the inlet for the third tundish configuration.

Figures 4(a) and 4(b) show the predicted flow fields in one half of the four-strand bare tundish system which represent the velocity fields in $XZ$ plane (central longitudinal vertical plane) at $Y/W=0$ and in $YZ$ plane (central transverse vertical plane) at $X/L=0$ (inlet stream) respectively. It can be clearly seen that the flow is highly turbulent near the inlet region of the tundish. The incoming jet hits the bottom of the tundish and very high velocity components are created in the bottom region along inlet-exit direction as is evident from the Fig. 4(a). The strong velocity field along the central plane, at which the exits are located, is responsible for short-circuiting in the inner strand of four-strand tundish. The liquid after striking the tundish bottom then partly moves up along the tundish side walls to the free surface, partly moving downstream in the direction of the exit and the rest recirculating back toward the incoming jet. The strong velocity fields thus created are responsible for reversal of flow towards the inlet stream and generated anticlockwise recirculatory flow at the right side of the inlet stream and may cause slag droplets entrapment. The overall C curves of both halves of the four-strand bare tundish are presented in Fig. 5. Both the C curves can be seen to be similar in their respective nature of variation. Each curve is characterized by two peak values of concentrations, one soon after the tracer injection and the second after some time. After attaining the second peak tracer concentration decreases continuously with time. Appearance of two peak values of concentration suggests that flow of the fluid be short-circuited of the tundish. Short-circuiting of the fluid is undesirable in tundish fluid system. It can also be seen that the overall C curve for one half (derived from two individual C curves lying on one half of the tundish) of the tundish is almost identical to the other. This indicates that the flow is symmetrical with respect to the inlet stream of the tundish. Moreover, these shows that due to symmetry flow, the two identical C curves of only one half can be taken as representative C curves for overall tundish (since the other pair can be deduced automatically). Thus the bare tundish is associated with considerable short-circuiting and large dead volumes (viz., Table 3), high turbulence by incoming stream jet and slag entrapments which are potentially detrimental to the floatation of non metallic inclusions.

Figures 6(a) and 6(b) show the predicted flow fields of longitudinal and transverse vertical planes of one half of
the four-strand tundish with dam. It can be seen that the Dam forces the fluid of the inlet stream to move towards free surface before it is transferred to the downstream side of the dam in the tundish. The region of the tundish upstream of the dam acts like a source for transfer of the kinetic energy of the inlet stream into the rest of the tundish. This is in contrast to bare tundish (viz., Fig. 4(a)) wherein energy is dissipated from the point of impingement of the inlet stream. Thus, the dam restricts the turbulence of the inlet stream within the inlet region as a consequence of which the fluid flow becomes uniform in the rest of the tundish. It can be further seen that the flow is surface directed; however, the flow having upstream the dam region is divided into two parts. One proceeds towards the inlet streams from the dam while another advances opposite and away from the inlet. Thus, anti-clockwisely rotating recirculatory flow appeared in the inlet region whereas clockwisely rotating recirculatory flow was created in the rest of the tundish region. The clockwisely rotating recirculatory flow having somewhat less intense turbulence and created in the other region of the tundish due to surface directed components of the fluid is not very suitable for production of clean steel (see Fig. 6(a)). It is noted that if any inclusion is entrapped into the inlet stream it is likely take very long time to float out, resulting into more accumulation of inclusions which would further increases the extent of contamination of steel. This recirculatory flow may prompt inhomogeneity (strand to strand variations) during the continuous casting multi-strand tundish systems. Figure 6(b) shows the predicted velocity fields of transverse vertical planes at different exit locations. It can be seen from the figure that the velocity field at the inner strand plane is upward direction whereas the flow is downward at the outer strand. It is also expected from the above conclusion that the use of a dam, the tundish bottom area and its surrounding wall refractory where the highly turbulent incoming inlet jet directly strikes, may be damaged frequently. To overcome from the above shortcomings associated by a dam, the novel design of flow control devices are used in this study.

The flow fields of the four-strand tundish with TID with holes and dam combination are shown in Figs. 7(a) and 7(b). Figure 7(a) shows the flow field on longitudinal (XZ) plane of half tundish at Y/W=0 while the flow field on transverse vertical plane at X/L=0.49 (inlet) is shown in Fig. 7(b). As seen, the TID produces a strong upward flow in the immediate vicinity of the ladle shroud and in conjunction with the dam, prevents any significant fluid motion along the bottom wall of the tundish as is normally witnessed in bare tundish. This is a desirable feature of the flow, which limits short-circuiting. Also it is interesting to note here that the anti-clockwisely rotating recirculatory flow is totally eliminated at the inlet region in contrast to bare tundish as well as tundish with a dam and thus could lead to better inclusion removal. The corresponding flow pattern along the transverse vertical plane at different tundish length, X/L=0.49 (inner strand) and X/L=0.89 (outer strand) is shown in Fig. 8. The velocity fields of inner strand and outer strand are grossly seen to be identical, both qualitatively and quantitatively. This promotes the strand similarity i.e., minimum strand to strand variations during the casting process of multi-strand tundish systems. Also, the velocity field at the inlet plane shows the clockwisely rotating recirculatory flow, which is desirable for producing...
clean steel. It is also noted that these phenomena were totally absent in the bare tundish and much less pronounced in the tundish with a dam. Also, in Fig. 9, the effect of a combination of TID and dam on individual C curves of four-strand tundish system is shown. As seen from the figure, the combination of the TID and a dam improves the RTD parameters for both strands considerably.

Figures 10(a) and 10(b) show the plots of predicted turbulence kinetic energy contours along the longitudinal vertical planes with TID+dam at \( Y/W \) of (a) 0, and (b) 0.78. The dissolution of alloy will be very fast along with higher convective and diffusive mixing in this region and will get uniformly distributed in the rest of the tundish, which results strand homogeneity during the casting.

In the present study two different shapes of C curves were obtained as has been presented earlier for single-phase system. One of these curves is characterized by the presence of two peaks (see Fig. 5 for bare tundish) and other by a single peak value of concentration (see Fig. 9 for tundish with TID+dam). Based on preceding discussion on fluid flow and RTD therefore suggests that following four types of flow regions are associated with the tundish system:

- Short circuited region
- Plug flow region
- Mixed flow region and
- Dead flow region

The different associated tundish volumes are the direct indices of the metallurgical performance in the tundish system. The corresponding flow volumes interconnected with characteristics of C curves are shown in Figs. 11(a) and 11(b). The slow moving dead region with short-circuiting or bypass flow is presented in Fig. 11(a) where as Fig. 11(b) characterizes the flow having slow moving dead volume in which the fluid continuously exchanges with the fluid in the active region (a combination of plug and well-mixed flow volumes). The overall RTD parameters with associated volume fractions for different tundish configurations are listed in Table 3. The basis for systematic hydrodynamic analysis
of these parameters, however, has already been well presented in our earlier publication.\textsuperscript{20} It can be seen that the short-circuiting fractions are practically zero except for bare tundish. It can also be seen that the associated dead volume is significantly high for existing bare tundish. The application of dam improves the flow pattern and brings down the dead volume from 27 to 17\% and it is further decreased to 12\% with the TID+dam configuration as is clearly mentioned in Table 3. Both the short-circuiting and dead regions associated in tundish are undesirable feature and therefore we wish to maintain these phenomena as low as possible for sound tundish operations. The better mixing in the tundish occurs corresponding to higher well-mixed along with lower dead volume and therefore it is also expected for better temperature homogenization (e.g., tundish with dam configuration). On the other hand, higher the plug volume with minimum dead region shows the better inclusion removal in the tundish (for example, tundish with TID+dam configuration).

The measured and numerically predicted time averaged vertical and horizontal flow components are compared directly in Fig. 12 for two different bath heights (180 and 250 mm respectively) at liquid flow rate of 1.55\times10^{-4} m^3/s. In these, it is seen that both numerical prediction and experimental measurements suggest that relatively high velocity regions are contained in the vicinity of the shroud. Furthermore, far away from the port region, while horizontal components are somewhat pronounced (on a relative scale), the vertical component of the flow is practically zero. It is also evident from the computational results and experimental observations that the intensity of fluid motion in most of the flow domain is extremely sluggish. Figure 12 evidently confirm that the FLUENT based model is able to simulate the observed flow phenomena fairly realistically. Based on visual judgment it is considered that the extent of mismatch between predicted and experimental flow components in these figures is not any greater than 10\% or so. It has already been reported\textsuperscript{21} that the velocity fluctuations within the flow system are specially dependant and the assumption of an isotropic fluctuations from the $k$-$\varepsilon$ model formulation of FLUENT is found to be reasonable (having variations within 20–30\% with the anisotropic fluctuations) and is, therefore, not reiterated here.

A direct comparison between predicted and experimental overall C curves of four-strand tundish with TID+dam for the single-phase system is illustrated in Fig. 13. There, reasonably close correspondence between the two is readily evident. It is instructive to note here that convective mass transfer processes depend on the mean and the fluctuating velocity component not to pronouncedly (for example, mass transfer coefficient varies in proportion to $\tilde{u}^n$ and $\tilde{u}^m$, $n$ and $m$ both being much smaller than unity). Con-
5. Conclusions

A combined experimental and computational study was carried out to investigate the fluid flow and residence time distribution of the four-strand continuous casting tundish with and without flow control devices. To this end the PIV measurement and tracer dispersion experiments for water model tundish were conducted for a wide range of design and operating parameters. A FLUENT based mathematical model incorporating the $k$-$\epsilon$ turbulence was applied to predict time averaged and tracer dispersion profile in the water model tundish. The results predicted by the model were verified against equivalent experimental measurements. The following conclusions have been drawn from the present study:

1. In the present study two different shapes of $C$ curves are obtained: one shape of the curve is characterized by two peak values of concentration and other characterized by single peak value of concentration. The first type of $C$ curve appears due to short-circuiting flow in the bare tundish.

2. Use of dam eliminates the short-circuiting phenomena in the tundish: one at the inlet region and other for away from the inlet. The former is undesirable, due to possibility of inclusion entrapment. Beside this, the bottom of the tundish is frequently damaged due to striking of the highly turbulent inlet jet.

3. Among all types of configurations applied in the present study, a combination of TID with holes+a dam work reasonably found to be an optimum configuration of the four-strand tundish. A superior strand similarity is achieved in this configuration.

4. Predicted time averaged horizontal and vertical components agreed within ±10% with the experimentally derived ones.

5. The measured and predicted tracer dispersion curves agree extremely well for single-phase system in the presence and absence of flow control device.

6. The general agreement between experimental and observations and predicted results is a function of the mathematical model configuration. The use of an extremely fine grid ($\text{viz.}$, nearly $150,000$ cells in a domain size of $1 \times 0.5 \times 0.3$ m) and the proper locations of the fluid-inlet plane ($\text{e.g.}$, farther upstream of the actual port), together with appropriate boundary conditions at the free surface ($\text{e.g.}$, symmetry type) and tundish outlet ($\text{e.g.}$, standard outflow), are critical as far as obtaining reasonable computational results are concerned.

REFERENCES