PROCEEDINGS : CAMME-1996
©NML JAMSHEDPUR; pp.99-119

An improved turbulence model for flow simulation in a continuous casting tundish for slabs

S. K. DAS

National Metallurgical Laboratory, Jamshedpur - 831 007, India.

ABSTRACT

A three dimensional mathematical model has been developed based on finite volume technique with improved (K-ε) two equation turbulence model to simulate fluid flow phenomena in steel continuous casting (CC) tundish for slabs. Considering the fact that at locations near the walls and in the vicinity of flow control devices of the tundish, where the liquid velocities are relatively low, the inclusion particles may move to the laminar sub-layer and stick to the wall. An attempt has been made to resolve the near wall region in a better way by applying the concepts of low Reynolds number turbulence model. The model is designed to be executable in computer systems with minimum 32 MB RAM. Computed results describing the 3-D flow field and the residence time distribution (RTD) during the steady state operation are presented. The mathematical model is then verified with the published data obtained from full scale water model. The various indexes of residence time such as t_{mean} *and* t_{peak} *obtained from the RTD curves are used to analyze the effect of various combinations of flow control devices on the removal of inclusion. The results show that the characteristics of fluid flow behaviour within a tundish can be varied considerably with the use of flow control devices incorporating near wall effects.*

INTRODUCTION

Continuous casting operation involves pouring of molten metal from the tundish into a water cooled mould, so arranged that partially solid metal of desired shape is drawn as a strand from the bottom of the mould. The strand is cooled in the secondary and tertiary cooling zones and when completely solidified is normally cut to length and reheated prior to rolling. A tundish is an intermediate vessel that taps the molten metal from the ladle and distributes it to the continuous casting mould. In order to produce clean metal/alloy, the tundish is also used as a reactor for removal of non-metallic inclusion in addition to its conventional role as reservoir and distributor. For the inclusions to be removed from the molten metal, the particles have to float to the top and be trapped by the surface slag layer. From the standpoint of tundish design and operation, the residence time of the molten metal has to be prolonged and the path lines of the non-metallic inclusion particles should favour their entrapment. Therefore, the velocity and pressure distribution and distribution of turbulence field in the molten metal strongly influences the removal of inclusion particles. To study such matters, detailed velocity and turbulence fields are required, these being specific to a given tundish design, metal flow rate and other parameters. It is believed that by properly inserting flow control devices such dam and/or weir, and optimizing submerged nozzle parameters, the flow of molten metal can be controlled and thus most of the inclusions can be removed.

There have been, in recent years a number of studies on fluid flow and/or inclusion separation behaviour for tundish arrangement, using physical (water models[1-6]) and/or mathematical model[7-12]. Using a full scale water model tundish, Kemeny *et.al.*[1] carried out a simple fluid dynamic analysis of tundish flows with the aim of improving steel cleanliness by maximizing fluid retention time. They observed that the flow patterns were improved using combinations of flow modification devices and the minimum fluid retention time can be increased. Tanaka and Guthrie[5] developed a probe to detect non-metallic inclusions in aqueous systems. Nakajima[6] extended the use of such probes to molten steel systems. Both investigators analyzed the separation behaviour of inclusion particles in terms of "tank reactor model". A schematic view of the full scale water model tundish is shown in Fig. 1 which will be taken as the reference model for the present mathematical simulation.

Szekely and El-Kaddah[7] carried out numerical prediction of 3-D using tundish fluid flow and RTD curves with and without Flow Control Devices (FCD) using the commercial PHOENICS code and Lai et.al.[8] carried out both mathematical and physical modelling of three dimensional fluid flow in a symmetrical twin strand tundish and compared on to the other. Also, he and Sahai[9] performed a computation of fluid flow in tundish under the condition of sloping side walls and compared the effects of these with vertical walls in terms of flow patterns and RTD curves. Take and Ludwig[11] solved the inclusion transport equation for particles taking into account their specific buoyancy, convection and turbulent dispersion, again using the commercial PHOENICS code. More recent contribution by Joo and Guthrie[10], Sahai and Chakraborty[13] studied the thermal stratification phenomena during steady state operation and during sequence casting with thermal cycles generated by cooling steel in teeming ladles[14].

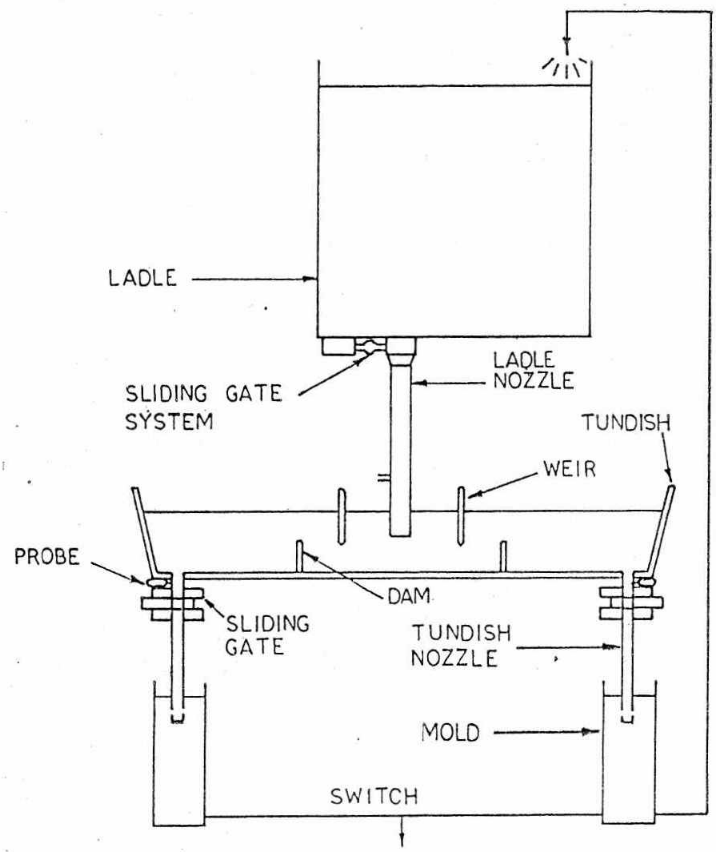

Fig. 1 : Schematic diagram of the tundish model in casting process

Most of the major steel companies still study the change of fluid flow and inclusion particle behaviour with or without flow control devices using isothermal physical modelling in large plexiglass water models. Flow visualisation, residence time distribution studies and detection of inclusion particles in such physical models certainly provide useful information. However, such full-scale physical modelling often requires expensive equipment and significant time and effort and most of the data generated are, in general, proprietary in nature. On the contrary, computational modelling of 3-D fluid flow are becoming less expensive and are widely applicable for prediction of flow field, inclusion flotation behavior and temperature distribution. These predictions based on mathematical modelling is useful for determining the best design of the vessel with respect to the dimensions, shape and placement of flow control device for a given set of operating parameters. However, their validity in general terms has not yet been clearly demonstrated either through direct comparisons with physical models or plant data. Paucity of reliable data is also one of the major constraint for such validation.

The structure of the melt flow field, flow characteristics, inclusion behaviour and RTD values are greatly influenced by the field values of turbulence and velocity, not only in the recirculation zone, but also in the near wall region. In addition, if flow separation and reattachment occurs, the complexity of the flow field also significantly increases. At location near the walls of the tundish where the liquid velocities are relatively low, the inclusion particles may move to the laminar sub-layer and stick to the walls and thereby removed from the system[12-14]. It is of significant importance to model accurately the flow in the near wall region (the low-Re region) for better inclusion transport predictions. Low Reynolds number (Low-Re) models have been devised in such way that they solve for flow, concentration and temperature variables in the near wall region and the equations reduce to the standard (K-ε) model equations in the region away from the wall. The low Re models should be able to resolve the near wall region better[20-24]. To evaluate accurately the turbulence coefficients, which influences the mechanism of inclusion and momentum transport in the tundish melt flow accompanying flow separation and reattachment, it is indispensable to predict the turbulent flow field with reasonable accuracy.

In all the previous studies, the conventional model of turbulence to predict the turbulent recirculating flow field often accompanying flow separation and reattachment has been employed[15,16,17]. Though, the (K-ε) model is quite widely applied as turbulence prediction model for diverse applications and also quite useful, some major problems still remain unresolved. Many of the fluid flow situation of practical interest are turbulent, including flows encountered in CC tundish have one or more recirculation zone. Calculation of turbulent flows with the (K-ε) model involves employment of wall function as the boundary condition on the solid wall. However, their application to recirculation region is open to question and doubtful. Furthermore, in the event of flow separation and reattachment, the conventional (K-ε) model may not be able to predict the flow field accurately. On the hand, the conventional (K-ε) models have been significantly improved in recent years[20-27], such that they may work even in the vicinity of wall as well as flow control devices incorporating the near wall effects, which are known as "low-Reynolds number (K-ε) models".

It is, therefore, the objective of the current work to determine the flow field and inclusion transport predictions by using the low-Re number turbulence model as compared to conventional wall function model which does not explicitly deals with the near wall effects of turbulence as effectively as the former. The low Re-model of Lam and Bremhorst is applied to account for the near wall effects.

FORMULATION

Mathematical Model

For the analysis of fluid flow phenomena in the tundish of a slab continuous casting process certain assumption were made to ease the complexity of the task. First of all, the actual tundish has wall as shown in Fig. 2(a). In this study, the tundish is assumed to be rectangular as shown in Fig. 2(b). It is also assumed that the existence of the surface slag layer does not significantly influence the flow pattern of molten steel. The melt in the tundish is isothermal.

In order to describe fluid flow turbulence properties and inclusion (mass) transport phenomena in the continuous casting steel making tundishes, the governing partial differential equations requiring numerical solution are the equations of continuity, momentum and mass conservation in three dimensional, incompressible cartesian co-ordinate system in ensemble averaged form presented in cartesian tensor from:

Continuity Equation

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad \ldots \quad 1$$

Naviers-Stokes' Equation

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial P}{\partial x_i}\mu_{eff}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + \rho g_i \qquad \ldots \quad 2$$

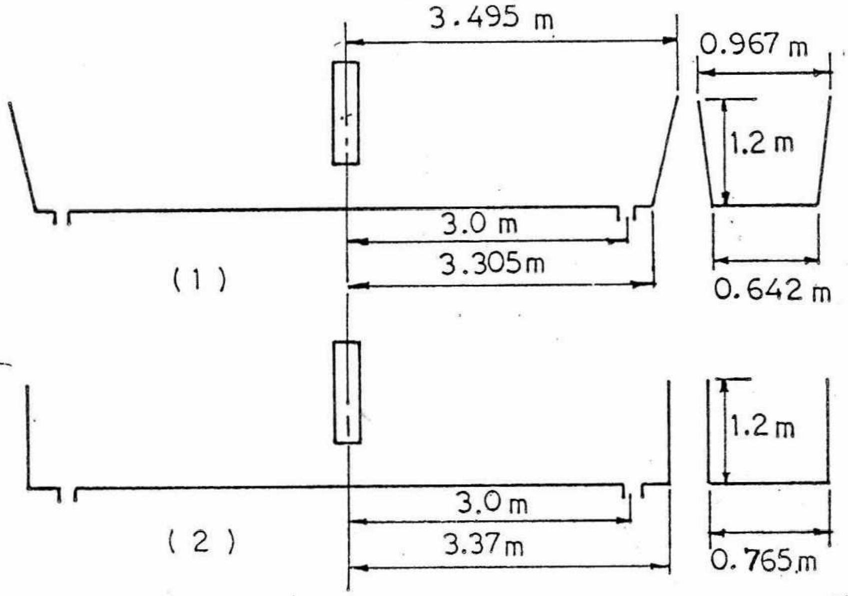

Fig. 2 : (a) Geometry and dimension of an actual tundish.
(b) Geometry and dimensions of the tundish analyzed in this simulation

Turbulence Modelling Equations

For modelling turbulence, the two equation (K-ε) model[15,17] are being used. There, turbulence is expressed by two transport equations for the turbulent kinetic energy K and its dissipation rate ε. The relation between the turbulent viscosity μ_t and these two characteristics of turbulence is:

$$\mu_t = C_D \rho K^2/\varepsilon \quad \ldots \quad 3$$

The governing equations for K and ε are,

$$\frac{\partial}{\partial x_i}\left(\rho\, u_i\, K - \frac{\mu_{eff}}{\sigma_k}\frac{\partial K}{\partial x_i}\right) = G - \rho\epsilon \quad \ldots \quad 4$$

$$\frac{\partial}{\partial x_i}\left(\rho u_i \epsilon - \frac{\mu_{eff}}{\sigma_\epsilon}\frac{\partial \epsilon}{\partial x_i}\right) = (C_1\, \epsilon G - C_2 \rho \epsilon^2)/K \quad \ldots \quad 5$$

Where, the effective viscosity comprises the laminar and turbulent component

$$\mu_{eff} = \mu + \mu_t \quad \ldots \quad 6$$

The viscous stress generation terms, G, are given by:

$$G = \mu_t \frac{\partial u_j}{\partial x_i}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) \qquad \ldots \quad 7$$

The five empirical constants appearing in equations (3) through (5) were adopted from (K-ε) model constants and given in Table-1.

Tabel - 1: (K–ε) model constants

C_1	C_2	C_D	σ_K	σ_ε
1.430	1.92	0.09	1.00	1.30

The mathematical model of inclusion transport employs a standard mass conservation equation describing the spatial distributions in mass fraction of fine particles within a fluid control volume. Thus, in the presence of steady flow field, the mass transport of inclusion particles can be expressed according to the species conservation equation of the form (neglecting Stokesian terminal rising velocity of the particle).

$$\frac{\partial C}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j C) = \frac{\partial}{\partial x_j}\left(\Gamma_{e\,c}\frac{\partial C}{\partial x_j}\right) \qquad \ldots \quad 8$$

where, $\Gamma_{e,c}$, the effective diffusivity can be written as:

$$\Gamma_{e,C} = \frac{\mu}{\sigma_C} + \frac{\mu_t}{\sigma_{t.c}} \qquad \ldots \quad 9$$

In this study $\Gamma_{e,c}$ is assumed to relate to the effective dynamic viscosity μ_{eff} in the following manner.

$$\frac{\mu_{eff}}{\mu} \approx \frac{\Gamma_{e.c}}{\Gamma} \qquad \ldots \quad 10$$

It may be noted that the transient and convective terms balance the diffusive term to the right hand side of equation. (8). The dispersion of micron size fine particles in metallurgical processing vessels can be described by a 3-D convection diffusion type mass transport equation. The solution of such equation requires that the distribution of flow variables (u_j, j =1,2,3) and turbulence variables be known *a priori.*. Furthermore, if the ladle has an insulated top, the ladle melt temperature decreases only 5–10°C and is always hotter than the melt in tundish the flow inversion phenomena will not occur and the flow pattern can be maintained for complete casting period. Therefore, the effect of temperature can be ignored and the assumption of isotherm for the incoming ladle melt stream and the melt in the bulk of the tundish is acceptable. Under such situation, it is not required to solve the heat transport equation.

When the relationship between inclusion concentration and time, namely RTD curve, is known certain important information can be obtained. Generally, in model studies, the first instant of discovering inclusions at the outlet nozzle after an impulse of inclusion is injected into the tundish from the inlet nozzle is called the minimum residence time, t_{min}. The time when the concentration is maximum in the RTD curve is called the peak residence time, t_{peak}. Also the mean residence time, t_{mean} is defined and estimated with the following equation

$$t_{mean} = \frac{\int_0^{\infty} tC.dt}{\int_0^{\infty} C.dt} \qquad \dots 11$$

Larger t_{mean} indicates that on average the residence time of the molten metal is longer; this favours the removal of inclusion particles. In addition to longer residence times, desirable features of any tundish flow should include; minimum surface fluctuation, elimination of short circuit, surface directed flow and minimum dead volume.

NEAR WALL TREATMENT AND MODEL FOR TURBULENCE

The (K–ε) model is most useful for the prediction of various turbulent flows in view of both accuracy and computational efficiency. However, the application of wall functions to flows with multiple recirculation zones, flow separation and reattachment is questionable, though it has been often employed as a boundary condition at a solid wall in applying (K–ε) model to the practical problems. Recently, to overcome this difficulty, the (K–ε) model has been considerably improved so that it may work by itself in the vicinity of the wall without employing the wall functions. With these improved models, which are usually called the "Low-Reynolds number (K–ε) models", the accuracy in predicting such flow phenomena has been improved significantly. An evaluation of many such low-Reynolds number models was done by

Patel, Rodi and Scheurer[19].

Since, the (K–ε) model is not valid in the inner region very close to wall and FCDs, where, the turbulent Reynolds number is small, it is necessary to modify the model in some way in the region. A common way of doing this is to use wall functions to account for the viscosity affected region close to the wall. Wall functions are however, not valid very close to the wall ($y^+ \leq 40$). As described in detail in [28], wall functions relate velocity as well as turbulent kinetic energy K, and the dissipation rate ε in the first grid point adjacent to wall to the friction velocity and lean heavily on the assumption of local equilibrium of turbulence (production = dissipation) at this point. These assumptions are not generally valid in cases with strong recirculation and not in separated flow either. Close to the wall ($y^+ \leq 40$), there is no balance between production and dissipation (dissipation >> production) such that the assumption of local equilibrium is not valid in this region.

Low Reynolds Number Turbulence Models

In low-Reynolds number models, the transport equations for all the dependent variables, including the dissipation rate, are solved in the near-wall region as well. The equations in the low-Reynolds (K–ε) model, differ from their basic versions (eqns. (3),(4) and (5)) by the inclusion of functions f_D, f_1 and f_2 to modify the turbulent constants, C_D, C_1 and C_2 respectively. The two models frequently used in low-Reynolds number turbulence simulation are those proposed by Lam and Bremhorst[20] (referred to as LB) and Herrero[29]. The expressions for the damping functions and the physical basis are briefly discussed below and can be found in greater detail in [20] and [29].

Damping Functions

The expression for the eddy viscosity μ_t may be written as

$$\mu_t = f_D C_D \rho K^2/\varepsilon \qquad \dots 12$$

where, the f_D function attempts to model the viscous effects on the eddy viscosity and has a predominant influence on the model performance. Lam and Bremhorst suggest that this damping function has to depend on the dimensionless local Reynolds number Re_n and Re_t, which are defined as :

$$Re_n = \frac{K^{1/2} n}{\upsilon} \quad \dots 13$$

$$Re_t = \frac{\rho K^2}{\mu \epsilon} \quad \dots 14$$

A criterion for f_D relates to the behaviour in the fully logarithmic region where it must tend to unity if the parent high Reynolds number ~~model~~ is to be recovered. Hence, the damping effect of f_D is restricted to the viscous sub layer and the buffer region. The next two damping functions, f_1 and f_2 will appear in the source term of the transport equation for the dissipation rate and the turbulent kinetic energy transport equation remains unchanged. The transport equations of the turbulence model is given as follows:

$$\frac{\partial}{\partial x_i}\left(\rho u_i K - \frac{\mu_{eff}}{\sigma k}\frac{\partial K}{\partial x_i}\right) = G - \rho\epsilon \quad \dots 15$$

$$\frac{\partial}{\partial x_i}\left(\rho u_i \epsilon - \frac{\mu_{eff}}{\sigma_\epsilon}\frac{\partial \epsilon}{\partial x_i}\right) = (f_1 C_1 \epsilon G - f_2 C_2 \rho \epsilon^2)/K \quad \dots 16$$

the functions f_1 increases the magnitude of ε near the wall and finally, function f_2 is introduced primarily to in corporate low-Reynolds number effects in the distribution term of ε–equation. The physical basis for this is provided by various experiments in the final period of the decay of isotropic turbulence. The expressions of the damping functions in the two of the most frequently used models are given below in the Table – II

Table – II: Expressions for damping functions in Low-Re models)

Low-Re Model	f_D	f_1	f_2
Lam-Bremhorst	$[1-\exp(-0.0165\, Re_n)]^2 x (1+20.5/Re_t)$	$1+(0.05/f_D)^3$	$1-\exp(-Re_t^2)$
Herrero et.al.	$[1-\exp(-0.0066\, Re_n)]^2 x [1+500\, \exp(-0.0055 Re_n)/Re_t$	$1+(0.05/f_D)^2$	$1-(0.31\, D) \exp(-Re_t^2)$ where, $D= 1-0.7 \exp(-Re_n)$

In the present work, Lam and Bremhorst model has been used to model the near wall effects of turbulent flow phenomena in the tundish.

Boundary Conditions

Fluid Flow

At the tundish walls including dams, weirs, and the side walls of the ladle nozzle, the variation in the flow properties are steeper than within the bulk fluid. By introducing the near wall damping functions, f_D, f_1 and f_2 the general form of the two-equation model (K–ε) for high Reynold number flow has been retained. In the regions far away from the wall where the flow is fully turbulent, these functions assume the value of unity thus conforming to the standard high-Re version of (K–ε) model. Under the framework of low-Re turbulence model, the boundary conditions for ε at the wall used by Lam and Bremhorst[20] is given as $\epsilon_w = \upsilon(\partial^2 K/\partial n^2)$. Alternatively, a more convenient boundary condition tested by Patel *et.al.*[19] is $(\partial\epsilon/\partial n) = 0$. Therefore, the zero gradient boundary condition has been used at the wall for ε. The condition K = 0 at the wall has been used as the boundary condition for K throughout this work. For velocity components parallel to the wall, non-slip conditions were imposed at the wall. At the symmetry plane and the free surface boundary, the normal velocity components and normal gradients of all variables are set equal to zero. At the jet entry, the velocity components perpendicular to the free surface was calculated from the volumetric flow rate and the cross-sectional area of nozzle (Ladle shroud).

$$U\epsilon = \frac{Q}{A_{(nozzle)}} \qquad \dots \ 17$$

Similar boundary conditions are imposed at the tundish outlet nozzles. The inlet values of K, the level of turbulent kinetic energy, and ε, the rate of dissipation have been approximated by

$$K_\varepsilon = 0.01 U_\varepsilon^2 \qquad \dots \ 18$$

$$\epsilon_{inlet} = \frac{K_{inlet}^{3/2}}{R_{nozzle}} \qquad \dots \ 19$$

Inclusion Transport

In order to simplify the problem of inclusion flotation, the following assumptions have been made in mathematical formulation.

1) Particles are spherical, and surface tension of particles has no effect on float out velocity.

2) There was no modelling of any interactions and/or agglomeration/ coalescence phenomena between inclusion particles within the tundish. Although coalescence is an important factor in the entry region where turbulence is high, it is supposed that subsequent coalescence would be minimal in the quieter regions of flow.

3) For the motion of inclusion/fine particles (in the range of 20 to 150μm), the Stokesian terminal rising velocity is neglected.

4) The sidewalls and the bottom of the tundish as well as FCD are all nonwetting to inclusions within the melt. Again, this is a simplification of practical situation. Inclusions such as alumina being known to be highly ferrophobic and likely to precipitate in the regions of flow reversal.

5) All boundary surfaces are impervious to the inclusion. Mathematically, this corresponds to zero flux at all the boundary surfaces.

Numerical Solution Procedure

The fully elliptic 3-D governing partial differential equations for u_i K, ε and C were discretized using finite integral volume method employing a nonuniform grid, hybrid differencing sheme[27]. The whole set of equations was solved via. a semi implicit tridiagonal matrix algorithm marching scheme coupled with a Gauss-Siedal routine. The Semi Implicit Method For Pressure Linked Equation (SIMPLE) algorithm extended to curvilinear coordinate system was used to solve the pressure field, through simultaneous satisfaction of the continuity and momentum equation within each volume element. The detailed implementation of the numerical algorithms is given by Shyy *et.al.*[30,31].

Only a symmetrical half of the model tundish was considered for the present computations. The domain was divided into a grid structure of (20 vertical) X 80 (longitudinal) X 30 (transverse) in three respective orthogonal directions. The low-Re model tries to resolve the near wall rtegion. There is always an issue as to how many points would be sufficient to resolve the wall region? In other words what value of y^+

should we have for near wall cells? For the given number of cells (20X80X30) in the symmetric half of the tundish with resulting y^+ value is in the range of 2-12.5 for the present low-Re model. The computer runs for isothermal conditions have been carried out on HP-800/F20 mini computer of 32 MB RAM and 48 MHz clock speed.

RESULTS AND DISCUSSION

Because the internal configuration of a tundish affects it's flow field, favourable flow conditions for the removal of non-metallic inclusions can be achieved by properly inserting flow control devices in the tundish. In order to demonstrate how the model can help achieve insight into the flow behaviour of liquid in tundishes of various designs, several cases were analyzed by the model developed in this study. The tundish dimensions and operating conditions used in the modelling is given in Table III. The four different design and operating conditions have been evaluated in this study. They include no flow control devices, the placement of a weir-dam, the placement of a dam-weir-dam, the placement of dam-weir-dam with a hole in the second dam. The locations and the dimensions of the dam/weir and given in Table IV. Only half of the modelled tundish is considered for the simulation due to its axial symmetry.

Table – III: Tundish dimensions and operating conditions

Half length of tundish	3.37m
Density of simulant (water)	1000 kg/m^3
Open size of inlet nozzle	0.0049 m^2
Open size of outlet nozzle	0.0049 m^2
Casting speed	1.2 m/sec
Initial liquid level	0.9 m

The 3-D calculated flow patterns for the four different designs, namely, no flow control device, placement of weir-dam, placement of dam-weir-dam, placement of dam-weir-dam with a hole in the second dam are shown in Figs. 3-6. The RTD curves and the values of residence time for various cases are shown in Fig. 7 and Table 4 respectively. The following observations have been made after careful study of the calculated flow field. For the tundish without flow control devices, the flow field is quite violent as shown in Fig. 3. The entering stream flushes down to the bottom, spreads radially, and shows a strong recirculation near the inlet nozzle. The entering jet, entraining liquid from its surrounding, causes a reverse flow at approximately

Table – IV: Various tundish design considerations in simulation study (meters)

Design Case	weir-dam	Dam-weir-dam	Dam-weir-dam with a hole
Position of weir	0.85	0.85	0.85
Distance between bottom and weir	0.15	0.15	0.30
Thickness of weir	0.065	0.065	0.065
Position of dam	1.70	0.30, 1.70	0.30, 1.70
Height of dam	0.25	0.35, 0.25	0.35, 0.35
Thickness of dam	0.065	0.065	0.065
Open size of the hole	-	-	0.10 x 0.07

one third of the total tundish length away from the inlet nozzle and an upward flow near the walls. The forward flow near the bottom and the backward flow near the free surface form a very large circulation pattern. The flow field shows a sink like vortex behaviour near the exit nozzle. The RTD curve (Fig. 7) for this case rises first and has the shortest t_{min} in all cases (Table V). The t_{min} is believed to be closely related to the short circuiting of fluid flow in tundishes. This suggests that nonmetallic inclusions may not have an opportunity to float up and get trapped by the surface slag. The trend is also shown by the t_{peak} and t_{mean}. In fact, the flow field in this case is not desirable for removal of inclusions.

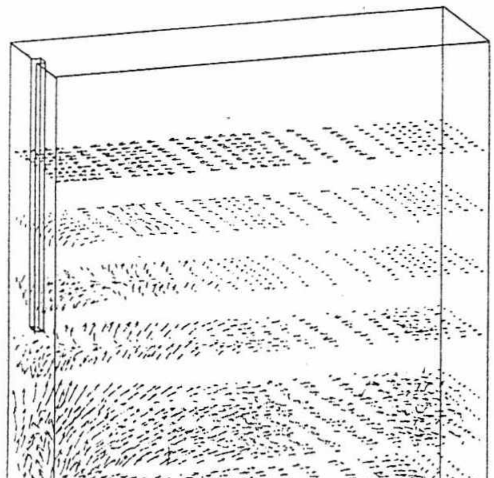

Fig. 3 : Isoparametric view of the calculated 3-D flow field in the tundish without flow control devices

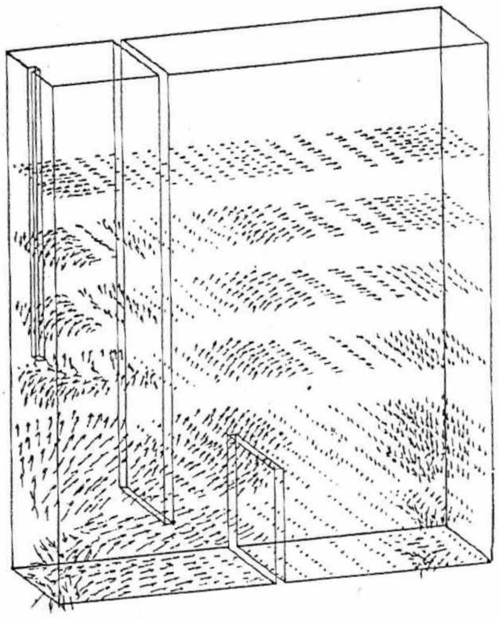

Fig. 4 : Isoparametric view of the calculated 3-D flow field: tundish with a weir and a dam

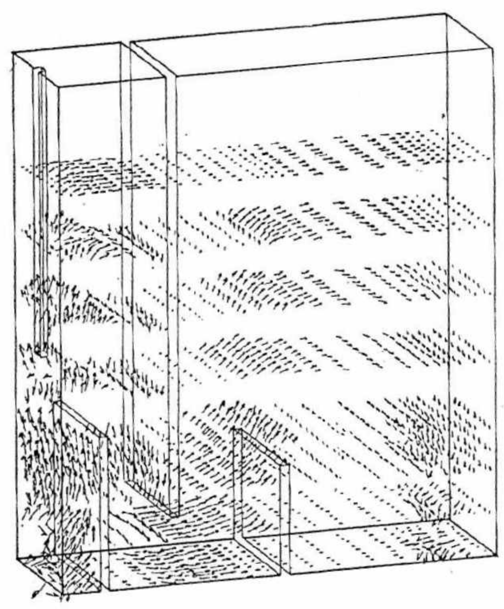

Fig. 5 : Isoparametric view of the calculated 3-D flow field: tundish with a weir and two dams

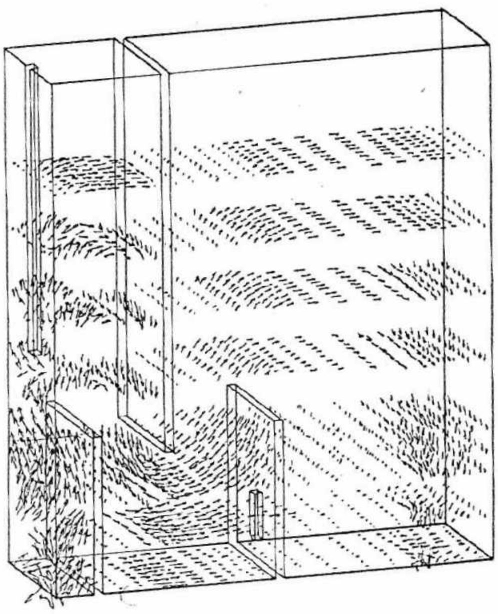

Fig. 6 : Isoparametric view of the calculated 3-D flow field: tundish with a weir and two dams (the second dam has a hole)

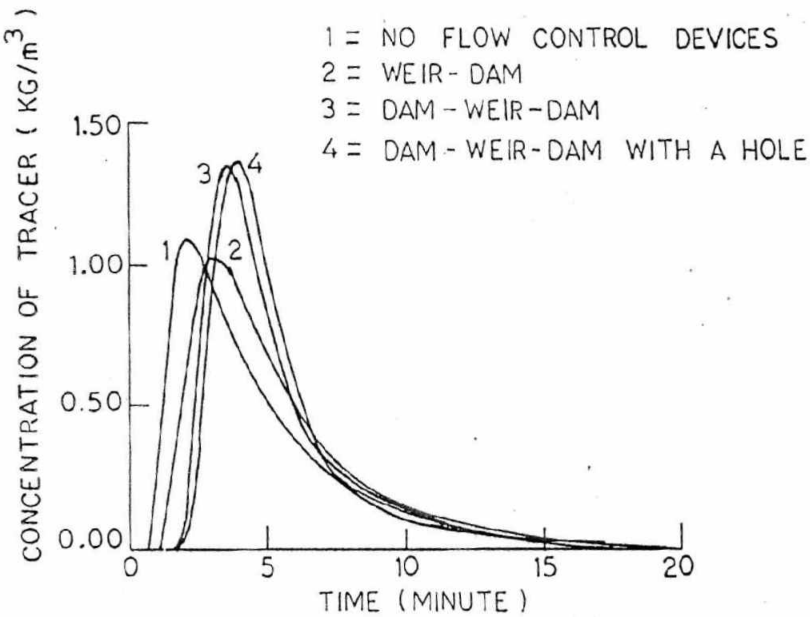

Fig. 7 : RTD curves calculated from the mathematical model for a tundish with various combinations of flow control devices

For the tundish with a weir and a dam, Fig. 4 shows that a strong recirculation was developed between the inlet and the weir. Therefore, the violent flow field can be restrained and a quiescent slag layer behind the weir can be obtained. The liquid was pressed to go under the weir and then forced to flow upward above the dam. This means that surface directed flow can be induced and the short circuit can be eliminated. The residence times can then be prolonged as shown in Table -V. These conditions promote good contact between the surface slag and the following melt and improve effectiveness of removing inclusions. In addition the dam is found to be beneficial at the end of the casting because it divides the skull into two sections, facilitating easier removal from the tundish. For the tundish with a dam-weir-dam, the dam near the inlet nozzle is observed from Fig. 5 to form a liquid pool, which can reduce the erosion of flowing melt on the tundish bottom underneath the ladle nozzle. The dams force the liquid to proceed upward and this surface directed flow can also be induced. The weir behind the first dam lengthens the flow path of the liquid. Because there are two dams in this case, it is expected that longer residence times can be obtained as shown in Table - V.

Table – V

(RTD values for various tundish operations) Mathematical Model

Residence time	Plain	Weir-dam	Dam-weir-dam	Dam-weir-dam-hole
t_{min}(sec)	36	56	87	94
t_{peak}(min)	2.27	3.15	3.70	4.03
t_{mean}(min)	4.81	5.49	5.57	5.60

Although the tundish with a dam-weir-dam can enhance inclusion particles' flotation, it has a dead zone behind the second dam and the residual steel is considerable at the end of the casting operation. To reduce the residual steel and alleviate the extent of the dead zone behind the second dam, it is desirable to make a hole in the second dam and raise its height. It can be observed from Fig. 6 that a higher dam deflected the stream more vertically towards the surface of the liquid pool. Therefore, this case can also induce surface-directed flow, eliminate short circuits, and prolong residence time even more, which can be seen in Table - IV. It can also be noted that the second dam with a hole does not shorten the minimum residence time as shown in Table - IV. Therefore, a dam with a hole not only forces the fluid to proceed upward and reduces the dead zone, but decreases the residual steel at the end of the casting. In addition, this case has the largest t_{mean} (Table-IV). It is therefore the most favourable tundish design among the cases investigated in this study. The broad trend of computed flow field and RTD values are in good agreement with the literature data[1,3,8,10].

CONCLUSION

In this study, a 3-D mathematical model has been developed based on finite volume Computation Fluid Dynamics (CFD) technique, in conjunction with an improved K–ε turbulence model, (known as Low Reynolds number turbulence model) to account for the near wall effects in the vicinity of the walls and FCDs. To account for the actual tundish geometry and dimension and near wall effects an uneven mesh system was used in the numerical scheme. It is observed that the convergence rate is found to be much slower than the high-Re model with wall functions. As an example, the wall function model needed about 600 iterations to yield a mass residual order of 10^{-3} to 10^{-4} whereas a typical low-Re model requires 4000-5000 iterations for the mass residual of same order. The mathematical model is found to be reliable by comparing it's results with measured data from a full scale water model of an actual tundish and other related mathematical models of full scale tundish. Then using the mathematical model, the flow behaviour of the liquid and non-metallic inclusions in the tundishes of various design were analyzed and the effects of placement of various combinations of FCDs (such as, weir, dam, dam with a hole) on the efficiency of non-metallic inclusion removal were evaluated. It is felt that, in general a high-or-low-Reynolds number model may not predict the experimental data for the entire range of Reynolds number for different design and operating conditions. For flow at Reynolds number lower than 12,000 the low-Re model performs better than wall function model. However, the Low-Re model tends to under predict the flow field at very high Reynolds number. The design of dam-weir-dam-hole configuration seems to be the most favourable design among the four cases studied. The tests of 3–D mathematical model with an improved model for turbulence demonstrate that it can provide a great deal of useful information which provides a better insight into the flow behaviour of molten steel in tundishes of various designs. It is also very flexible. The design and operating conditions can be easily changed and the effects of the changes on the outcome can be readily evaluated. With the aid of this design tool, optimization of tundish design criteria and operating conditions can be accomplished in a more systematic and scientific way to produce cleaner continuously cast steel.

LIST OF SYMBOLS

C	:	Species concentration
C_D, C_1 & C_2	:	Turbulence model contants
f_D, f_1 & f_2	:	Damping functions of low-Re model
G	:	Generation of turbulent kinetic energy
g	:	Gravity acceleration constant
K	:	Turbulent kinetic energy
n	:	Direction normal to wall
P	:	Pressure
Q	:	Volumetric flow rate of fluid
Re_n, Re_t	:	Dimensionless local Reynolds number
t	:	Time
u_i, u_j	:	Cartesian velocity components
x_i	:	Spatial cartesian coordinates
y^+	:	Dimensionless distance from the wall with reference to(K-ε) model

Greek Symbols

μ_{eff}	:	Effective dynamic viscosity
μ_t	:	Turbulent viscosity
ε	:	Dissipation Rate
$\Gamma_{e,c}$	:	Effective mass diffusivity
υ	:	Kinematic viscosity
σ_c	:	Laminar Schmidt number
$\sigma_{t,c}$	:	Turbulent Schmidt number
μ	:	Laminar viscosity
Γ	:	Laminar mass diffusivity

REFERENCES

[1] F. Kemeny, J.D. Harris, A. Mclean, T.R. Meadoweroft and J.D. Young, "Fluid flow studies in the tundish of a slab caster", *Proc. 2nd Process Technology*

Conf. Chicago. IL, ISS-AIME, Warrendale, PA, 1981, pp232-245.

[2] D.J. Harris and J.D. Young, "Continuous Casting", *ISS-AIME,* Warrendale, PA, 1983, Vol. 1 pp99-112.

[3] M. Hashio, M. Tokuda, M. Kawasaki and T. Watanabe, "Improvement of cleanliness in continuously cast slabs at Kashima steel works", *Proc. 2nd Process Technology Conf. Chicago. IL,* ISS-AIME, Warrendale, PA, 1981, pp65-73.

[4] J. Knoepke and J. Mastervich, "Water modelling at Inland Steel's No. 3 combination caster tundish", *Steelmaking Proc.,* Vol 69, 1986, pp777-788.

[5] S. Tanaka and R.I.L. Guthrie, *Proc. Japan - Canada Seminar on secondary steel making,* Tokyo, Japan, Dec. 1985, Paper No. C3.

[6] H. Nakajima, *Ph.D. thesis,* McGill University, Montreal, 1987, Canada.

[7] J. Szekely and N. El-Kaddah, "The mathematical modelling of three dimensional heat flow, fluid flow and turbulence phenomena in tundishes", *Steelmaking Proc.,* Vol. 69, 1986, pp761-776.

[8] K.Y.M. Lai, M. Sa Cudean, S. Tanaka, and R.I.L. Guthrie, "Mathematical modelling of flows in large tundish systems in steel making", *Metall. Trans. B*, Vol. 17B, 1986, pp449-459.

[9] Y. He and Y. Sahai, "The effect of tundish wall inclination on the fluid flow and mixing: a modelling study", *Metall, Trans. B,* Vol. 18B, 1987, pp81-92.

[10] S. Joo and R.I.L. Guthrie, "Heat flow and inclusion behaviour in a tundish for slab casting", *Can Metall. Q.,* Vol. 30 1991, pp261-269.

[11] K.H. Tacke and J.C. Ludwig, "Steel flow and inclusion separation in continuous casting tundish", *Steel Res.,* Vol. 58(6), 1987, pp262-269.

[12] R.I.L. Guthrie, S. Joo and H. Nakajima, "Mathematical models and sensors as an aid to steel quality assurance for direct rolling operations", *Proc. Int. Symp. on direct rolling and hot charging of strand cast billets, CIM,* Montreal, 1988, Pergamon Press, Elmsford, NY, Vol. 10, 1988, pp193-212.

[13] Y. Sahai and S. Chakraborty, "Importance of thermal modelling in continuous casting tundishes", *Proc. 10th Process technology Conf., Toronto, ISS-AIME,* Warrendale, PA, 1992, pp161-176.

[14] S. Joo : *Ph.D thesis,* McGill University, Montreal, Canada, 1989.

[15] B.E Launder and D.B. Spalding, "The numerical computation of turbulent flow", *Comp. Meth. Appl. Mech. Engg.,* Vol 3, 1974, pp269-289.

[16] B.E. Launder, "On the computation of convective heat transfer in complex turbulent flows, *ASME. J. Heat Transfer,* Vol 110, 1988, pp1112-1128

[17] W.P. Jones and B.E. Launder, "The prediction of laminarization with a two

equation model of turbulence", *Int. J. Heat Mass Transfer,* Vol. 15, 1972, pp301-314.

[18] C.G. Speziale "Analytical methods for development of Reynolds stress closures in turbulence Ann. Rev. Fluid Mech., Vol 23, 1991, pp107-157.

[19] V.C. Patel, W. Rodi and G. Scheuerar, "Turbulence models for near-wall and low Reynolds number flows : A Review, *AIAA J.* 23, Vol 23, 1985, pp1308-1319.

[20] C.K.G. Lam and K. Bremhorst, "A modified form of the K-ε model for predicting wall turbulence", J. Fluids Engng, 1981, vol 103, pp 456-460.

[21] Y. Nagano, M. Tagawa and M. Niimi,"An improvement of the K-e turbulence model (the limiting behaviour of wall and free turbulence, and he effect of adverse pressure gradient)", *JSME. J. Ser,* 1989, B-55, pp1008-1015.

[22] Y. Nagano and M. Tagawa, "An improved K-ε model for boundary layer flows", *J. Fluids Engg.,* Vol 112, 1990, pp33-39.

[23] Y. Nagano and M. Hishida, "Improved form of the K-e model for wall turbulent shear flows", *J. Fluids Engg.,* Vol 109, 1987, pp156-160.

[24] E.W. Adams and J.P. Johnston, "Flow structure in the near-wall zone of a turbulent separated flow", *AIAA J.,* Vol 26, 1988, pp932-939.

[25] T. Abe K. Kondoh and Y. Nagano, "A new turbulence model for predicting fluid flow and heat transfer in separating and reattaching flows - I. Flow field calculation", *Int. J. Heat Mass Transfer,* Vol 37, No. 1, pp139-151.

[26] M. Ciofalo and M.W. Collins., "K-ε predictions of heat transfer in turbulent recirculating flows using an improved wall treatment", *Heat Transfer,* B-15, 1989, pp 21-47.

[27] S.V. Patankar, "Numerical Heat Transfer and Fluid Flow", McGraw-Hill, New York, NY 1980.

[28] W. Rodi., "Turbulence models and their application in hydraulics", *International association of hydraulic research Monograph,* 1980, Delft.

[29] J. Herrero, F.X. Grau and J. Grifoll, "A near wall K-ε formulation for high Prandtl number heat transfer", *Int. J. Heat Mass Transfer,* Vol 34, No. 3, 1991, pp711-721.

[30] W. Shyy, S.S. Tong and S.M. Correa : Numerical recirculating flow calculations using a body-fitted coordinate system, Numer. heat Transfer, Vo. 8, 1985, pp99-113.

[31] M. Braaten and W. Shyy, "A study of recirculating flow computaion using body fitted coordinates", Consistency aspects and mesh skewness, *Numer. Heat Transfer,* Vol. 9, 1986, pp559-574.