Modelling and validation of fluid flow inside a dissipative ladle shroud and a continuous casting tundish

J Zhang¹,³, Q Liu¹, Z Xin¹, B Lu², J Zhang² and J Li²

¹State Key Laboratory of Advanced Metallurgy, University of Science and Technology Beijing, Beijing 100083, China
²HBIS Tangshan Iron and Steel Co. LTD, Tangshan, Hebei 063000, China

E-mail: zjsustb@163.com

Abstract. Fluid flow plays a significant role in the continuous casting of molten steel. In this paper, two Reynolds Averaged Naiver-Stokes (RANS) models and a Large Eddy Simulation (LES) model were comparatively employed to characterize the fluid flow inside a dissipative ladle shroud and a tundish. LES model was proved to be powerful to characterize the turbulence structure inside the ladle shroud. The effect of meshing density on the computational accuracy was considered using the LES model. The fine-mesh model can capture multiscale vortices inside the ladle shroud; while, the coarse-mesh model disables the LES to obtain the detailed flow information. The experiment of particle image velocimetry (PIV) was used to verify the flow field obtained by the LES model inside the tundish. The PIV and the LES results agree well in terms of flow pattern and velocity vector.

1. Introduction

In the continuous casting process, the flow field of molten steel is of significance to the quality of casting blanks [1]. The turbulent flow inside the ladle shroud and tundish is associated with the air absorption, slag entrapment and inclusion removal, which could deteriorate the production. The method of combining physical and mathematical model has been widely used to study the flow field. For instance, the Reynolds Average Naiver-Stokes (RANS) model combined with hydraulic simulation is the most widely used strategy [1,2]. For example, Chattopadhyay et al [2] investigate the effect of submergence depth of ladle shroud on the fluid flow inside the tundish using a RANS model. Zhang et al [3] studied the flow behaviour inside a five-strand tundish with ladle changeover operation using a RANS and VOF model. RANS method has the advantages of low computation cost, easy convergence, and can obtain the main flow information. However, RANS is unable to give an accurate prediction of turbulence due to the time-averaged turbulence information at all scales of the whole flow field [4]. Direct Numerical Simulation (DNS) is capable of capturing all scale eddies, but requires very fine mesh and extremely high computational cost, which make it unfeasible to be applied in engineering applications. Large Eddy Simulation (LES) sits in between the RANS and DNS models and solve the large-scale eddies directly using the Naiver-Stokes equations with modelled small-scale eddies. A filter equation is used to differentiate the scale of the eddies and the small ones are solved using a subgrid model, which allows an in-depth understanding of the flow structure [5]. However, the LES is still less used to study the fluid flow in the tundish and the fundamental mechanism is still unclear.

This study focuses the modelling and validation of fluid flow inside a dissipative ladle shroud
(DLS) using both mathematical and physical modelling. Two RANS models and a LES model were examined. The numerical results were compared with physical observations. This study could be useful to better understanding the flow structure during the continuous casting process.

2. Experimental

2.1. Mathematical modelling

2.1.1. Assumption. (1) Water, the fluid, was treated as incompressible continuum and its physical properties are fixed. (2) The flow was assumed to be isothermal in the ladle shroud. (3) The effect of argon injection and slag was neglected.

2.1.2. Governing equations. The flow inside the ladle shroud falls into turbulence. In RANS model, the instantaneous velocity and pressure are divided into the mean part and the fluctuating part. For example,

\[ U_i = \bar{U}_i + U'_i \]  

where \(\bar{U}_i\) is the time-average velocity and \(U'_i\) is the fluctuating velocity. The mean part is achieved by averaging over a time scale, while the fluctuating part is less considered. Thus, the equations of continuity and Navier-Stokes are expressed as follows.

\[ \frac{\partial \bar{U}_i}{\partial x_i} = 0 \]  

\[ \frac{\partial \bar{U}_i}{\partial t} + \frac{\partial \bar{U}_i}{\partial x_i} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu_{\text{eff}} \nabla^2 \bar{U}_i - \frac{\partial (u'_i u'_j)}{\partial x_j} \]  

In equation (3), \(u'_i u'_j\) indicates the Reynolds stress tensor, which can also be denoted as \(R_{ij}\) and solved following the Boussinesq’s hypothesis. The symbol of \(\nu_{\text{eff}}\), effective viscosity, is composed of the molecular (\(\nu_0\)) and kinetic viscosity (\(\nu_t\)). The above equations are not closed to solve the five unknown variables and different turbulence models have been developed to close one or more of the equations.

The \(k-\varepsilon\) model adopts the concept of Eddy-viscosity and solves the length and velocity scales in equation (3) using two equations. The kinetic viscosity is expressed as:

\[ \nu_t = C_\mu \frac{k^2}{\varepsilon} \]  

where \(k\) and \(\varepsilon\) are the turbulent kinetic energy and the dissipation rate respectively. These two values are solved by solving equations (5) and (6).

\[ \frac{\partial k}{\partial t} + \bar{U}_j \frac{\partial \bar{U}_i}{\partial x_j} = \nu_t \frac{\partial k}{\partial x_i} + \nu_t \frac{\partial \bar{U}_i}{\partial x_j} \left( \frac{\partial \bar{U}_j}{\partial x_i} + \frac{\partial \bar{U}_i}{\partial x_j} \right) - \varepsilon + \frac{\partial k}{\partial x_i} \left[ (v + \nu_t) \frac{\partial \varepsilon}{\partial x_i} \right] \]  

\[ \frac{\partial \varepsilon}{\partial t} + \bar{U}_j \frac{\partial \varepsilon}{\partial x_j} = C_{\epsilon_1} \frac{\varepsilon}{k} \nu_t \frac{\partial \bar{U}_i}{\partial x_j} \left( \frac{\partial \bar{U}_j}{\partial x_i} + \frac{\partial \bar{U}_i}{\partial x_j} \right) - C_{\epsilon_2} \frac{\varepsilon^2}{k} + \frac{\partial \varepsilon}{\partial x_i} \left[ (v + \nu_t) \frac{\partial \varepsilon}{\partial x_i} \right] \]  

Reynolds Stress Model (RSM) solves the transport equations for each of the terms in the Reynolds stress tensor, which shows clear advantage over the \(k-\varepsilon\) model on flow with high degrees of anisotropy, such as curved flow and rotational flow.

\[ \frac{D R_{ij}}{D t} = P_{ij} + D_{ij} - \varepsilon_{ij} + \Pi_{ij} + \Omega_{ij} \]
Rather than averaging the time-dependent turbulence, LES model employs the concept of filtering, i.e., decomposing the turbulent eddies into large scales and small scales. The large-scale eddies are directly solved using the Naiver-Stokes equation while the contribution of small ones to turbulence is modelled using the so-called subgrid-scale (SGS) model. In FLUENT package, the spatial filtering is enabled based on the mesh size [5]. In the LES model, the Naiver-Stokes equation can be obtained by substituting the Reynolds stress tensor into the SGS stress tensor, which is defined as:

\[ \tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \]  

(8)

Boussinesq’s hypothesis is used to again to model the SGS stress. However, the turbulent viscosity requires to be modelled and the Wall-Adapting Local Eddy-viscosity (WALE) model was used in the current study, which is believed to be reasonable and accurate in flows associated with complicate geometries [6]. The \( \nu_t \) is given as [5]:

\[ \nu_t = \rho L_s^2 \frac{(S_{ij} S_{ij})^{3/2}}{(S_{ij} S_{ij})^{5/2} + (S_{ij} S_{ij})^{5/4}} \]  

(9)

where \( L_s \) is the mixing length for sub-grid scales, defined as:

\[ L_s = \min(\kappa d, C_w V^{1/3}) \]  

(10)

where \( d \) is the distance from the cell center to the closest wall; the von Kármán constant \( \kappa \) is 0.418. \( C_w \) is chosen to be 0.325, which has been confirmed to yield satisfactory results for a wide range of flow in FLUENT [5].

2.1.3. Computational domain, boundary conditions and computational strategy. The structures of the DLS and the small tundish are shown in figures 1 and 2 respectively. Their dimensions are listed in table 1. Water was selected as the fluid and its properties are shown in table 1 as well. The DLS comprises three enlarged chambers and a bell-shaped outlet. The effect of mesh density was considered, and the domain was meshed with coarse, fine and very fine grids as shown in figure 1.
The DLS is partly immersed into the tundish with a depth of 36.5 mm. The flow rate is 30 L/min. The tundish is a small scaled tundish with simplified structure, as the main purpose of the current study is to investigate the fluid flow inside the ladle shroud.

The boundary conditions of the domain are defined as follows.
- **Inlet:** a constant inflow velocity (0.708 m/s) profile was applied at the nozzle inlet;
- **Outlet:** a fixed pressure of 0 Pa (relative to the ambient) was applied at the outlet of the tundish;
- **Top surface and walls:** free-slip boundaries with zero normal velocity were used at the top surface. All walls of the domain were considered to be no-slip, and the wall boundary domain was handled using the Werner-Wengle formulation [7].

The governing equations were discretized with 2nd-order central differencing scheme for convection terms. Implicit Fractional Step Method was employed to realize the velocity-pressure coupling. The time integration was performed with 2nd order implicit scheme. The time-dependent LES modelling was started by a steady state calculation using the k-ε two-equation model to help the convergence.

### 2.2. Physical modelling

A water model was established to investigate the fluid flow inside the domain with the same flow parameters and validate the mathematical results. A schematic illustration of the experiment set-up is shown in figure 3. A rotor was placed under the ladle shroud to measure the velocity magnitude. The LES results were also compared with measured results using PIV from the literature.

![Figure 3](image)

**Figure 3.** The velocity vectors inside the middle chamber of DLS with different mesh densities using LES: (a) coarse, (b) fine and (c) very fine.

### 3. Results and discussion

#### 3.1. The effect of mesh density

The spatial mesh density is associated with the filtering operation in the LES model. It is shown in
figure 3(a) that small-scale eddy is barely captured due to the large spatial size. The vector is apparently denser in figure 3(c) compared with the results in figure 3(b). However, figure (b) basically shows equivalent ability to show the morphology of the eddies.

Vorticity ($\zeta$) is one of the most important properties of fluid flow. The curl of fluid velocity is the vorticity of flow field, which is defined as:

$$\zeta = \nabla \times \vec{V} = \begin{vmatrix} \hat{i} & \hat{j} & \hat{k} \\ \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \\ u & v & w \end{vmatrix}$$

The unit of vorticity is $s^{-1}$. In the continuous casting process, the internal flow is mostly spiral due to the irregular of geometric structure, the viscous fluid of molten steel and the wall boundary layer. Vorticity is randomly distributed and composed of fluctuating vorticity of different scales. Thus, the vorticity magnitude is an important index to evaluate the fluid flow inside the DLS and the tundish. Figure 4 shows the contours of vorticity magnitude inside the DLS with different mesh densities. From the coarse to fine mesh, LES tends to capture finer turbulence scale, more obvious spin of eddy current, and larger vorticity. The maximum vorticity under the coarse grid is 150 $s^{-1}$, which can capture large vortexes with poor accuracy. The maximum vorticity under fine and very fine grids is 480 and 700 $s^{-1}$, respectively. Compared with these three vorticity contours, the fine grids can basically capture all turbulent structures.

In addition, the time consumptions to proceed 1.0 s calculation are 7 min, 18 min and 30 min for the coarse, fine and very fine meshes respectively. Thus, the fine mesh is selected to carry out the LES and RANS simulation considering a relatively acceptable computational cost.

![Figure 4](image1.png)

**Figure 4.** The contour of vorticity magnitude inside the DLS with different mesh density: (a) coarse, (b) fine and (c) very fine.

![Figure 5](image2.png)

**Figure 5.** The contour of vorticity magnitude inside the DLS using different models: (a) $k-\varepsilon$, (b) RSM and (c) LES.

**3.2. The results of different turbulent models**

Figure 5 shows the contour of vorticity magnitude inside the DLS using different models in the dissipative long nozzle calculated by SKE, RMS and LES models. It is noticed that the two RANS models can only capture a small number of vortexes, and the maximum vorticity value is less than 130 $s^{-1}$. This is mainly determined by the calculation method of Reynolds average, which results in the averaged vortexes with large vortexes and small vortexes. The LES results presented in figure 5(c)
show that the model can capture vortices of different scales, and the maximum vorticity value captured is close to 600 s$^{-1}$.

Figure 6 shows two typical velocity vectors selected in the middle chamber. It can be seen that the instantaneous flow characteristics of RANS barely change at different times, and cannot show the random flow and eddy. These largely limit the RANS model to study the detailed flow structure inside the DLS with satisfactory accuracy. However, the computational efficiency of these two models is much higher than that of LES, and the convergence is relatively easy. Besides, the overall flow trend in the nozzle can be obtained.

These results confirm the advantages of LES in transient simulation. Compared with RANS model, LES is able to capture turbulent information of fluid more comprehensively, which is conducive to in-depth discussion of rotational motion in the domain. From the viewpoint of fluidynamics, the fluid flow in the DLS is a kind of non-equilibrium complex turbulence and LES has good adaptability to this kind of flow.

3.3. The fluid flow inside the tundish

To validate the numerical results, the velocity magnitude is compared at two points under the DLS (P1 and P2), as shown in figure 7. P2 is 10 mm below the point P1 to facilitate the measuring. The rotor
measured the distribution of velocity at the outlet of 200 groups, and the results show that the fluctuation range of velocity is 0.081–0.585 m/s; its average velocity at P2 is 0.269 m/s. LES findings show the fluctuation of P1 within 10 s. The velocity fluctuation range of LES results is large, ranging from 0.059 to 0.754 m/s, and its average velocity is 0.314 m/s. The average velocity measured at P2 is 14.3% lower than that at P1. This difference is mainly due to the fact that the measurement area and time step are much larger for the rotor than those of the LES. The rotor velocity measurement is 7 homogenized in a small area, so the measured value is relatively small as a whole. However, both of the measured results reflect that the velocity fluctuation at the outlet of the DLS is severe.

The LES model has also been validated by comparing simulation results with PIV-measured results of Ken Morales-Higa’s work [8]. 2D PIV can capture the overall flow of fluid in a certain surface or region. It can be seen from the figure 8 that the two flow patterns are quite similar. After flowing out of the DLS outlet, the flow stream first shifts to the right-hand side, and then divides into two sub-streams near the bottom of the tundish, which flow to the left and right sides respectively. The left branch moves to the side wall of the tundish and goes up, creating a large circulation flow. The sub-stream on the right of the tundish quickly enter the whole pool and mix with the fluid in the tundish. In this flow mode, the velocity at the pouring region is within the range of 0.3 to 0.5 m/s, and it dissipates further after entering the whole tundish. Therefore, the comparison with the previous experimental results shows that the numerical simulation model basically matches well with the physical results. However, comparing the numerical and physical results, the location and size of the vortex cannot match completely, which is mainly caused by the complex and unpredictable nature of turbulence.

Figure 8. Comparison of the fluid flow in the pouring zone using (a) LES, and the measured results using PIV from [8].

4. Conclusions
Both numerical and physical simulations were conducted to investigate the fluid flow inside a DLS and a tundish. It is found that the LES simulation is sensitive the mesh density and a fine mesh is selected by comparing the vorticity magnitude and computational cost. The LES results successfully capture multiscale vortices inside the ladle shroud, while the two RANS models fail to capture the vortex due to the time-averaged operation in the simulation. The numerical and physical results match well in terms of flow pattern and velocity distribution; however, the velocity magnitudes are slightly different due to the complexity of turbulence. The LES model can be used to study the flow structure inside the ladle shroud and the tundish.

Acknowledgments
The funding of Independent Subject of the State Key Laboratory of Advanced Metallurgy (No. 41619001) is highly acknowledged.
References
[1] Chattopadhyay K, Isac M and Guthrie R 2010 ISIJ Int. 50 331-48
[2] Chattopadhyay K, Isac M and Guthrie R 2011 Ironmak. Steelmak. 38 398-400
[3] Zhang H, Fang Q, Luo R, Liu C, Wang Y and Ni H 2019 Metall. Mater. Trans. B 50 (online)
[4] Zhang J, Yang S, Li J, Yang W, Wang Y and Guo X 2015 ISIJ Int. 55 1684-92
[5] FLUENT6.3-Manual ANSYS Inc Lebanon NH 2007
[6] Nicoud F and Ducros F 1999 Flow Turbul. Combust. 62 183-200
[7] Piomelli U and Balaras E 2002 Annu. Rev. Fluid Mech. 34 349-74
[8] Morales-Higa K Guthrie R Isac M and Morales R 2013 Metall. Mater. Trans. B 44 63-79