Performance of turbulence models for flow prediction in a mould of continuous steel caster

J Gregorc¹, A Kunavar¹ and B Šarler¹,²

¹ Laboratory for Fluid Dynamics and Thermodynamics, Faculty of Mechanical Engineering, Aškerčeva 6, SI 1000 Ljubljana, Slovenia
² Laboratory for Simulation of Materials and Processes, Institute of Metals and Technology, Lepi pot 11, SI 1000 Ljubljana, Slovenia

E-mail: bozidar.sarler@fs.uni-lj.si

Abstract. The flow of molten steel in a mould during continuous casting process is nowadays most often studied using numerical approach. With such an approach, appropriate numerical tools have to be selected to achieve physically sound results. Within this work we focus on performance of different turbulence models for the case of casting of steel billets with single SEN port oriented in vertical, downward direction. Only fluid dynamics aspect of the casting process is considered, while all other phenomena are disregarded at this stage. Simulations were performed with commercial CFD package ANSYS Fluent with k-ε, k-ω and SAS turbulent models. The geometric model, material properties and process conditions are closely matched to those from our experimental water model studies, which is typically not done. Hence, the difference between the experiments and the numerical simulation originates only due to turbulence model performance. The experimental PIV data from the water model experimental system is used to benchmark the performance of various turbulence models. The models are compared based on lateral dispersion of the liquid jet exiting submerged entry nozzle (SEN) and formation of recirculation zones around the SEN. The results show that while all three models are capable of capturing general flow features, only the SAS model predicts finer turbulent structures and reasonably captures the flow behaviour in temporal domain.

1. Introduction

Steel has been a source of survival and development throughout the human history and continues to be one of the most important materials at our disposal. Steel production continuous to increase even in the first decade of the 21st century. Steel can be recycled and reused in new products, making it one of the earliest sustainable materials in modern economy. With time the steel production underwent huge improvements in sense of productivity, product quality and lately also energy consumption as well as other elements of sustainability [1].

The advances in computer capabilities and understanding of physics in the recent decades gave unique ability to study, predict and control processes happening in environment and/or conditions where measurements and trial & error type of approach is nor possible nor safe. As a consequence, the numerical simulations of continuous casting of steel casting are extremely important [2–6].

The meshless local radial basis function collocation method (LRBFMC) for solution of incompressible turbulent flow, based on k-epsilon model, was applied for numerical solution of solidification in continuous casting of steel in [5]. The method proved to be efficient and was upgraded also to account for electromagnetic stirring [7, 8].

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd
In the past, RANS approach to turbulence modelling was typically used [9, 10]. With increased computational power, however, the so-called “scale resolved simulations”, like LES, are becoming more and more affordable [11, 12], however, the computational effort for such a simulation is still huge. The aim of our current work is to assess the capabilities of two RANS turbulence models and a SAS (scale adaptive simulation) model that belongs to so-called “RANS-LES hybrid” class of models. It is still capable to resolve at least some of the scales while requiring less computational effort. ANSYS Fluent code, without any alterations to turbulence models, was used to predict the flow structures in the geometric model of water experiment for continuous casting. Majority of the published papers, both experimental and numerical, focus on slab casting using SENs with multiple side ports of different shapes and sizes [6, 9, 10, 13]. In our case, however, casting of steel billets using SEN with single vertical downward port is considered. The results are compared with the measurements conducted on experimental rig without any geometric scaling and by using the same material properties for both, numerical simulation and experimental investigation.

Based on the assessment of the fit of the simulations with our own experimental data an informed decision is made in the present paper on which turbulence model to implement into own LRBFMC code in order to improve its prediction capabilities.

2. Numerical setup and procedure
The goal of this numerical study is to assess the performance of three different turbulence models; the first belonging to the k-ε family, the second to the k-ω, and the third is a SAS model.

2.1. Governing equations
Flow in the computational domain is assumed to be turbulent, incompressible, two-phase (gas and liquid with a free surface in-between), isothermal and unsteady. For such a case mass conservation reduces to the conservation of volume fraction (given by equation (1)) and momentum conservation reduces to the equation (2).

\[ \frac{\partial \alpha_L}{\partial t} + \nabla \cdot (\alpha_L \mathbf{V}) = 0 \]
\[ \alpha_L + \alpha_G = 1 \]  
\[ \frac{\partial}{\partial t} (\rho \mathbf{V}) + \nabla \cdot (\rho \mathbf{V} \mathbf{V}) = -\nabla p + \mathbf{V} \cdot \mathbf{t} + \rho \]  

The free surface at the top of the domain is described by VOF model using an algebraic approach for interface tracking. The surface tension effects were not considered. Material properties, e.g. density, viscosity for two phase system were computed for each cell based on the volume fraction by using equation (3).

\[ \rho = \alpha_G \rho_G + (1 - \alpha_G) \rho_L \]
\[ \mu = \alpha_G \mu_G + (1 - \alpha_G) \mu_L \]  

2.1.1. Turbulence modelling. Three different RANS/URANS turbulence models were considered in this study: The Model 1 (M1): Realizable k-ε model [14] with Enhanced wall functions (Rk-ε EWT), the Model 2 (M2): SST (Shear Stress Transport) k-ω model [15] and the Model 3 (M3): SAS model [16]. All belong to two “so-called” equation classes of turbulence models where two additional transport equations are solved, for turbulence kinetic energy (k) and its dissipation rate (ε) / specific dissipation rate (ω). The general form of transport equation for k (equation (4)) is similar for all three models. G_k and G_ω are production terms for k due to mean velocity gradients and buoyancy, respectively and Y_k is the destruction term for k. The G_b term is equal to 0 in case of M2 and M3 models.

\[ \frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - Y_k \]
Transport equations for turbulent kinetic energy dissipation rate $\epsilon$ (equation (5)), in case of M1 model and turbulent kinetic energy specific dissipation rate $\omega$ in case of M2 and M3 models, are presented in equations (5), (6) and (7). The difference between these equations are that $G_{bc}$ term (dissipation production due to buoyancy) is only present in $\epsilon$ equation and $D_{\omega}$ (cross-diffusion modification) term is only present in both $\omega$ based models to allow for the blending of the standard $k$-$\omega$ and the standard $k$-$\varepsilon$ approach.

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + G_\epsilon + G_{bc} - Y_\epsilon$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_\omega} \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_\omega} \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + Q_{SAS}$$

The equations for M2 and M3 models are essentially the same, except for one term, named $Q_{SAS}$. This term introduces von Karman length scale into the system of equations; allowing the model to adjust its turbulence length scale to the already resolved scales. This effectively means that the model is capable to resolve the additional turbulence structures in “LES-like” manner, down to the grid spacing, making this model a kind of a hybrid between the RANS and the LES models. For a detailed information on the source term definitions and the implementation of the models used, is the reader referred to [14–17].

2.2. Discretization
The geometry model is a 1:1 copy of the industrial caster used by our industrial partner (figure 1). At the same time, it is identical to the geometry of experimental test section of the water model. To reduce the total number of the computational cells, the upper part (100 mm) of the mould above the free surface, was cut, while still maintaining enough space to avoid numerical artefacts due to the moving interface boundary between the gas and the liquid.

Block structured, hexahedral approach was used to build a computational mesh on geometric model. O-grid type meshing was adopted to ensure high quality cells between the cylindrical SEN and the hexagonal mould (see figure 2). Special attention was devoted to area at the bottom of the SEN (as can be seen from figure 2(b) and figure 2(c)) where the inlet boundary is created, assuring smooth transition of the mesh in all three directions. The mesh details on the top and the bottom of the domain are given in figure 2(a) and figure 2(d), respectively. All together 43 blocks were needed to construct the mesh with $2,287,800$ hexahedral cells. The mesh was built in accordance with the guidelines for RANS turbulence modelling without any wall functions, ensuring $y^+\sim 1$ on the walls.

The mesh independence was performed on three different meshes with constant refinement factor in accordance with the best CFD practice. A simplified Richardson’s extrapolation reveals that the relative discretization error for the mesh with $2,287,800$ cells is below 5%. Therefore, this mesh is considered in all the calculations that follow. Further refinement would also increase already extensive computational efforts. The simulations are run on 24 core server (AMD Opteron8439SE 2.8 GHz) with computation time ranging from 3-5 days.

2.3. Boundary conditions and solution procedure
There were three different boundary patches defined in the discrete model: inlet, at the bottom of SEN, outlet at the top of the domain and outlet at the bottom of the domain. The inlet was defined as velocity inlet, where velocity was prescribed as vector components in form of a profile obtained by separate simulation of fluid in the SEN. Pressure outlet was used for both, the domain top and the bottom. At the top, gauge pressure was set to 0 Pa, on the bottom the value of gauge pressure was such that the free surface was kept at the correct position. The materials were water and air. Their properties are listed in table 1.
Figure 1. The basis for the geometric model.

Figure 2. Overview of the mesh on the critical locations.

The temporal discretization is done using the second order implicit formulation. SIMPLE algorithm is used for pressure-velocity coupling. Spatial discretization is based on using Least Squares Cell Based, PRESTO! and compressive scheme for gradients, pressure and volume fraction, respectively. In case of momentum equation and turbulence quantities, second order upwind scheme is used.

Equations are solved by using ANSYS Fluent 18.2 which incorporates the finite volume framework. The time steps are between $10^{-3}$ and $5 \times 10^{-3}$ s. Minimal simulated time is determined by expression $t_{\text{min}} = 3L_{\text{domain},Y} (V_{\text{inlet-mean}})^{-1}$, where $L_{\text{domain},Y}$ is the domain length in vertical direction and $V_{\text{inlet-mean}}$ is the mean velocity at the inlet. The convergence is checked at every time step by using the absolute convergence criteria with the values of globally scaled residuals set to $10^{-4}$.

3. Experimental setup and methods

Without any doubt, the experimental data on flow behaviour in the mould are of key importance as they provide an insight into flow behaviour (essential in the design phase of numerical models) and can be used to validate the developed numerical tools.

Experimental test section considered for this work is based on industrial caster for steel billets used by Štore Steel company in Slovenia. The test section is made of two parts; the mould and SEN. The water model of the mould is made from 8 mm thick acrylic. It is slightly curved in vertical direction and has a square cross section with the side 190 mm (figure 3(a)). The SEN has inner and outer diameter of 35 mm and 65mm, respectively (figure 3(b)). The liquid exits in vertical direction.

Experimental rig is schematically shown in figure 4. The system operates as follows: water exits from tank (3) into the pump (4), that is set to the desired flowrate controlled with flowmeter (5). At the top, water enters the SEN and then the mould. Two valves are placed in series at the exit from the mould. The ball valve (7) is used during starting, stopping and filling procedures, membrane valve (8), on the other hand, is used for precise positioning of the moving surface at the top of the mould.
3.1. Experimental conditions and process similarity
Experiments are performed at isothermal ambient conditions; hence, all thermal effects are disregarded. Care was taken when performing study of flow similarity between the industrial conditions using molten steel and laboratory tests, conducted with water to ensure that results would be relevant. Investigation was performed at industrially relevant casting speed 1.4 m/min. Based on true (industry) casting speed and mould cross section, the volume flow rate of the material passing through the mould was estimated. It is worth noting that the material shrinkage due to solidification cannot be considered. With known geometry of the mould, volume flow rate and material properties (see table 1) at typical casting temperatures, Reynolds number for molten steel was estimated. Using this value and material properties for water (see table 1) at ambient conditions, the necessary flowrate of water was calculated. The mean velocity at the end of SEN was estimated to be 1.02 m/s.

Table 1. Material properties for similarity analysis.

| Material       | Density (kgm$^{-3}$) | Dynamic viscosity (kgm$^{-1}$$^{-1}$) |
|----------------|----------------------|--------------------------------------|
| Water (liquid) | 0.9982×10$^{-3}$     | 1.003×10$^{-3}$                      |
| Air (gas)      | 1.2250×10$^{-3}$     | 1.820×10$^{-5}$                      |
| Steel (liquid) | 6.8839×10$^{-3}$     | 5.318×10$^{-3}$                      |

3.2. PIV measurements
Particle Image Velocimetry (PIV) measurements are performed by using 2D PIV system from Dantec Dynamics. For seeding 10 μm rhodamine particles (encapsulated rhodamine) is used. They are illuminated by dual pulse laser capable of delivering 65 mJ per pulse at wavelength 532 nm. The $\Delta t$ between the pulses is set to 1.7 ms. The particle reflections are recorded with 4 Mpix high sensitivity, double exposure camera. The system operates at 15 Hz. The duration of the data acquisition is 10 s, providing a total of 150 velocity fields per measurement. The vector fields and data post processing are structured in DynamicStudio software.

4. Results
The first task is to estimate the discrepancy between the numerical prediction and the measurements. This is done by comparing the contours of the velocity magnitude in $x$-$y$ (figure 5) and $y$-$z$ (figure 6) planes, both passing through SEN axis. The results presented are instantaneous, taken from 214th second of simulations and one randomly selected time in measured time series. As seen in both figures, it is
clear that all three models are capable of capturing basic shape of the main jet exiting the SEN. Velocity field is more or less steady in case of M1 and M2 models. The shapes of the velocity magnitude contours are symmetrical to SEN axis. For M3, on the other hand, the flow doesn’t really become steady, but rather resembles true “turbulence like” behaviour that seems to be far more realistic and closer to what was experimentally observed.

![Velocity magnitude contours in x-y plane at z = 0 (middle of the SEN) for M1 (a), M2 (b), M3 (c) and experiment (d).](image)

**Figure 5.** Velocity magnitude contours in x-y plane at z = 0 (middle of the SEN) for M1 (a), M2 (b), M3 (c) and experiment (d).

There are certain regions in experimental contour plots, that show high local velocity that seems to be unphysical/unexpected; e.g. region around the interface or bottom corners. These artefacts are products of light scattering from the free surface at the top or result of poor laser light intensity at the limits of the observed area (view window).

![Velocity magnitude contours in y-z plane at x = 0 (middle of the SEN) for (a) M1, (b) M2, (c) M3 and (d) experiment.](image)

**Figure 6.** Velocity magnitude contours in y-z plane at x = 0 (middle of the SEN) for (a) M1, (b) M2, (c) M3 and (d) experiment.

The plots of y velocity components over the mould in x direction at z = 0 were made to further investigate the match between the numerical and the experimental results. The profiles presented in figure 7 are taken from two vertical positions: y = -50 mm (50 mm below SEN bottom, part a) and y = -150 mm (150 mm below SEN, part b) using the same temporal settings as in figure 5 and figure 6. The overall agreement between the results is acceptable, considering the fact that the experimental data and M3 data is changing from time step to time step, hence it is practically impossible to align the results in the temporal domain. On both presented locations are the maximal velocity values for all three models in range with the experimental data. The same can be concluded for widening of the jet cone. In the region of high velocity, the experimental results show significant velocity fluctuations. It is clear, that from the three turbulence models being tested, only M3 is capable of predicting at least some degree of these fluctuations.

![Velocity magnitude contours in y-z plane at x = 0 (middle of the SEN) for (a) M1, (b) M2, (c) M3 and (d) experiment.](image)
Figure 7. Profiles of $v_y$ in $x$ direction at $z = 0$ and (a) $y = -50$ mm and (b) $y = -150$ mm.

Figure 8. Turbulent structures represented as iso-values of $\lambda_2 = -0.15$ s$^{-2}$ along with the velocity magnitude contours on $x$-$y$ plane at $z = 0$ m for (a) M1, (b) M2 and (c) M3.

The results presented here conclusively show that M3 model can describe jet unsteadiness and its behaviour better than the rest two models used in this study. The fact that the general characteristics of the jet, like maximum velocity or velocity profile in the jet are predicted reasonably well even with simpler models, makes it difficult to “pick a winner”. This is especially true if one also considers computational time and resources needed to perform M3 simulation compared to RANS. However, there are scenarios where RANS approach simply isn’t enough. The study of the inclusions path (to estimate where the inclusions would accumulate) for example, would require to resolve at least some part of the turbulence structures (at least a few different length scales) as they would most certainly affect where inclusion would go. M3 as a hybrid RANS-LES model, capable of doing that, as can be seen from figure 8, where turbulent structures are represented iso-values of $\lambda_2$ parameter.

Within this study, the flow in the mould was assumed to be pure hydrodynamic problem. In case of flow in the caster at industrial conditions one also has to consider other physical phenomena of solidification that are coupled with the turbulence. The assessment of the performance of these models, when coupled also with the energy equation, EM field, and possibly solidification would be beneficial for proper prediction of the transport phenomena in continuous casting of steel.
5. Conclusions
Within this paper a performance of three different turbulence models is tested for the case of water flow in a mould of continuous steel caster for billets. With similarity analysis boundary values were estimated to assure hydrodynamic similarity between the flow in the industrial caster and the water model. The simulations were performed by using CFD package ANSYS Fluent 18.2. Results with M1, M2, and M3 models are presented and compared to PIV experimental data from the water model. The key findings are:

- All three models are capable of predicting general flow features in a reasonably good agreement with the experimental data.
- With M1 and M2 models the simulations reach steady state (with minor fluctuations) even if simulations are running in a transient manner.
- With M1 and M2 models is the shape of the velocity contours of the primary jet symmetrical to SEN axis.
- In case of M3 model, the steady state solution is not found, the flow around SEN axis is not symmetrical and the velocity fluctuations within the primary jet never fade out, which agrees with experimental observations.
- Model M3 is computationally the most expensive of the three, followed by M2 and M1 that are quite comparable.
- For anticipated further study of the interaction between the inclusions and the fluid flow is the M3 model to pick from the three.
- Future work needs to be done to test the performance of this models also when turbulence is coupled with other physical phenomena.

Acknowledgments
The authors gratefully acknowledge support by Slovenian Research Agency (grants P2-0162 and L2-9246), and Štore-Steel company (www.store-steel.si).

References
[1] Honeycombe R and Bhadeshia H K D H 1995 Steels: microstructure and properties (London: Edward Arnold)
[2] Šarler B, Vertnik R and Maček M 2017 Simulation of continuous casting of steel with EM stirring 9th ECCC European continuous casting conference - ECCC 2017 (Vienna) pp 903–12
[3] Šarler B, Vertnik R, Lorbiecka A Z, Vušanović I and Senčič B 2014 BHM Berg- und Hüttenmännische Monatshefte 159 99–107
[4] Šarler B, Vertnik R, Lorbiecka A Z, Vušanović I and Senčič B 2012 IOP Conf. Ser.: Mat. Sci. and Eng. 33 012021
[5] Vertnik R and Šarler B 2009 Int. J. Cast Met. Res. 22 311–15
[6] Thomas B G 2018 Res. Int. 89 1700312
[7] Vertnik R, Mramor K and Šarler B 2019 Eng. Anal. Bound. Elem. 104 347–63
[8] Košnik N, Vertnik R and Šarler B 2014 Mater. Sci. Forum 790–791 390–5
[9] Hershey D E, Thomas B G and Najjar F M 1993 Int. J. Numer. Methods Fluids 17 23–47
[10] Thomas B G and Najjar F M 1991 Appl. Math. Model. 15 226–43
[11] Yuan Q, Thomas B G and Vanka S P 2003 ISSTech 2003 (Conf. Proc.) 1 141–156
[12] Liu Z Q, Qi F S, Li B K and Jiang M F 2011 Adv. Mater. Res. 402 432–435
[13] Cho S-M, Thomas B and Kim S-H 2018 Metall. Mater. Trans. B 50 52–76
[14] Shih T-H, Liou W W, Shabbir A, Yang Z and Zhu J 1995 Comput. Fluids 24 227–38
[15] Menter F R 1994 AIAA J. 32 1598–605
[16] Menter F R and Egorov Y 2010 Flow, Turbul. Combust. 85 113–38
[17] ANSYS 2015 User Guide for ANSYS 18.2 (Cotonsburg, PA: ANSYS Inc.)