Engineering Analysis with Boundary Elements 45 (2014) 45–61

Contents lists available at ScienceDirect

Engineering Analysis with Boundary Elements

journal homepage: www.elsevier.com/locate/enganabound

Solution of a continuous casting of steel benchmark test by a meshless method

R. Vertnik [a,b], B. Šarler [b,c,d,*]

[a] Štore Steel, d.o.o., Železarska cesta 3, SI-3220 Štore, Slovenia
[b] University of Nova Gorica, Vipavska 13, SI-5000 Nova Gorica, Slovenia
[c] Institute of Metals and Technology, Lepi pot 11, SI-1000 Ljubljana, Slovenia
[d] COBIK, Tovarniška 26, SI-5270 Ajdovščina, Slovenia

ARTICLE INFO

Article history:
Received 12 June 2012
Accepted 20 November 2012
Available online 26 February 2014

Keywords:
Continuous casting of steel
Solidification
Turbulent flow
Mixture continuum model
Meshless numerical method
Radial basis function
Local collocation
Benchmark

ABSTRACT

This paper solves a recently proposed industrial benchmark test (Šarler et al., 2012 [1]) by a meshless method. The physical model is established on a set of macroscopic equations for mass, energy, momentum, turbulent kinetic energy, and dissipation rate in two dimensions. The mixture continuum model is used to treat the solidification system. The mushy zone is modeled as a Darcy porous media with Kozeny–Karman permeability relation, where the morphology of the porous media is modeled by a constant value. The incompressible turbulent flow of the molten steel is described by the Low-Reynolds-Number (LRN) k–ε turbulence model, closed by the Abe–Kondoh–Nagano closure coefficients and damping functions. The numerical method is established on explicit time-stepping, collocation with multiquadrics radial basis functions on non-uniform five-nodded influence domains, and adaptive upwinding technique. The velocity–pressure coupling of the incompressible flow is resolved by the explicit Chorin's fractional step method. The advantages of the method are its simplicity and efficiency, since no polygonisation is involved, easy adaptation of the nodal points in areas with high gradients, almost the same formulation in two and three dimensions, high accuracy and low numerical diffusion. The results are carefully presented and tabulated, together with the results obtained by ANSYS-Fluent, which would in the future permit straightforward comparison with other numerical approaches as well.

1. Introduction

Continuous casting [1,2] (Fig. 1) is the most common process in the production of steel. The process starts by pouring the molten metal into the water cooled mold, where cooling intensity is high enough to solidify steel around the inner surface of the mold and generates the solid shell with molten metal in the center of strand. After several minutes, the strand is pulled into the secondary cooling system, which contains water spray systems with much smaller cooling intensity compared to the internally water cooled mold, and rollers, which support and guide the strand up to the end of the casting machine. The quality of the cast product (round or square billets, blooms or slabs) depends mainly on the process parameters in the mold region, where complex physical phenomena occur (Fig. 2). The liquid metal, poured with a high velocity from the submerged entry nozzle (SEM) into the mold, produces turbulent flow with several re-circulating zones. Large heat fluxes, extracted from the mold, are a consequence of very high flow rates of the cooling water in the mold channels. It is impossible to measure temperature and velocity field inside the strand due to very high temperatures of steel and inaccessibility of the region during the process. Respectively, the numerical models [3] help us to better understand the casting behavior and to further improve and optimize the process, particularly in such experimentally sophisticated situations. Various numerical methods have already been used to simulate the described problem, such as Finite Volume Method (FVM) [4–8], Finite Element Method (FEM) [9], and more advanced meshless method, like the Local Radial Basis Function Collocation Method (LRBFCM) [10–12].

There is a continuously developing need for benchmarking of numerical models and methods – from the theoretical as well as from the applied points of view. The benchmarking is usually done in two parts. The verification part confirms the proper numerical solution (Are we solving the equations correctly?) and the validation part (Are we solving the right equations?) confirms the proper response of the numerical model regarding the experimental evidence.

Only very recently, different computational models (different governing equations, combined with different numerical methods)

* Corresponding author.
E-mail addresses: robert.vertnik@store-steel.si (R. Vertnik), bozidar.sarler@ung.si (B. Šarler).

http://dx.doi.org/10.1016/j.enganabound.2014.01.017
0955-7997

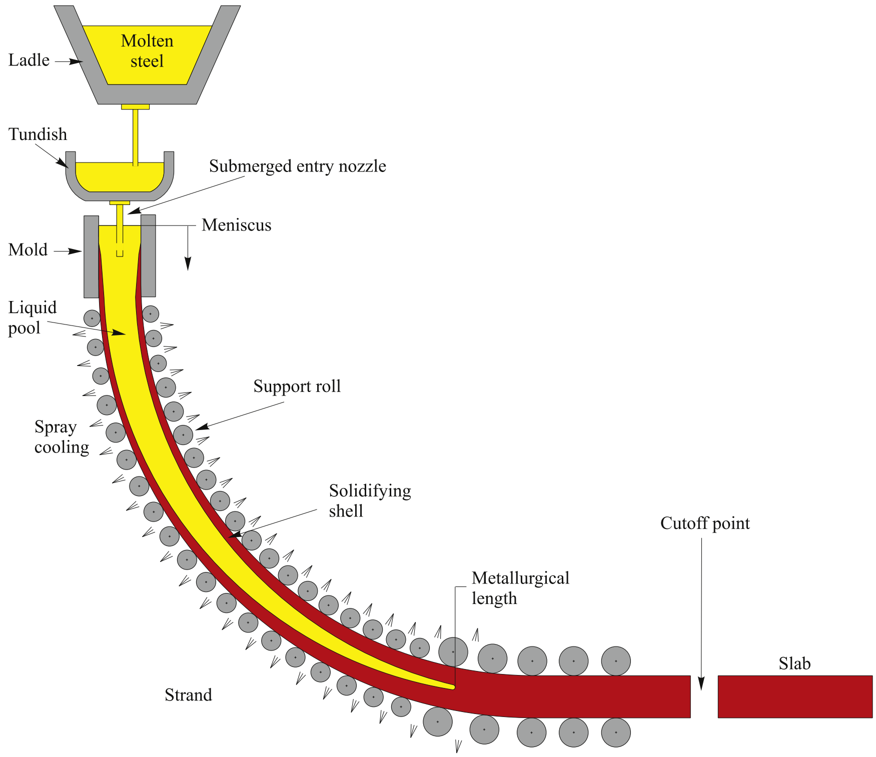

Fig. 1. Scheme of the continuous casting process.

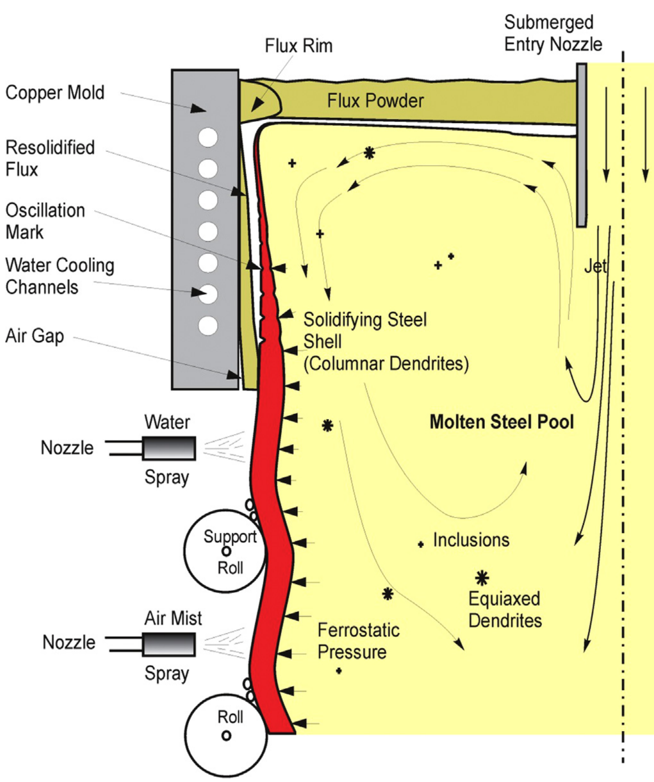

Fig. 2. Physical phenomena in the mold.

have been compared to assess the typical flow situations in continuous casting process [13] and their relation to model experiments. The solidification simulations are usually tested on a sequence of benchmarks that follow NAFEMS-type heat conduction simulations [14], convective diffusive cases with phase change [15], natural convection cases [16], solid–liquid specific cases with freezing of pure substance [17,18] and solidification of binary alloy [19,20]. However, until now, there is a lack of benchmark tests that would enable verification of numerical methods for solving transport phenomena in continuous casting of steel in a systematic approach. There are no systematic well documented benchmark tests currently existing even for such a simple continuous casting models like the slice model [21,22]. In this paper, a simple benchmark test for the numerical solution of the continuous casting of steel, proposed in [1] has been recalculated by LRBFCM and FVM. The main aim of the test is to compare the results of different numerical methods of the same physical model.

The original benchmark test [1] proposes a physical model that considers turbulent fluid flow with electromagnetic forces, solidification, and macrosegregation. In the present paper we consider only turbulent fluid flow and solidification. The solution procedure is based on explicit time discretization and fractional step method [23] for velocity–pressure coupling. Turbulent flow is modelled by the two-equation eddy-viscosity model with the low-Re corrections. The meshless LRBFCM is used for the spatial discretization. The comparison is made with the results obtained by the commercial software ANSYS Fluent, which is based on the FVM. The objective of this exercise is to test the ability of different numerical methods and algorithms to reasonably agree on a solution and to produce a reference numerical solution of the given problem for further comparisons.

2. Governing equations

Consider a connected domain Ω with boundary Γ occupied by a liquid–solid phase change material experiencing the solidification phenomena and turbulence. The material is described with the temperature dependent specific heat at constant pressure $c_{\wp}$ of the phase $\wp$ ($\wp = S$ for solid phase and $\wp = L$ for liquid phase), thermal conductivity $\lambda_{\wp}$, and the specific latent heat of the solid–liquid phase change h_m. The density ρ is in the present simple model assumed to be constant and equal for both phases, i.e. $\rho = \rho_S = \rho_L$. The liquid phase