Numerical modelling of the MHD flow in continuous casting mold by two CFD platforms ANSYS Fluent and OpenFOAM

Alexander Vakhrushev[a], Zhongqiu Liu[a,b], Menghuai Wu[a], Abdellah Kharicha[a], Andreas Ludwig[c], Gerald Nitzl[d], Yong Tang[d], Gernot Hackl[d]

a. Christian Doppler Laboratory for Advanced Process Simulation of Solidification and Melting, University of Leoben, Austria
b. School of Metallurgy, Northeastern University, Shenyang, 110819, China
c. Chair of Simulation and Modeling of Metallurgical Processes, Department of Metallurgy, University of Leoben, Austria
d. RHI Magnesita, Austria

ABSTRACT: *Electromagnetic brake (EMBr) technology is widely used to control the turbulence flow in the continuous casting mold. Experimental studies and numerical simulations are commonly employed nowadays to investigate the phenomenon and adjust the EMBr technologies. The aim of this study is to compare the commercial software ANSYS Fluent and the open-source CFD package OpenFOAM in order to verify their capability on modelling the magnetohydrodynamics (MHD) in the turbulent flow. Two mentioned CFD platforms are verified and compared based on the performed liquid metal experiment at a laboratory-scale slab caster (mini-LIMMCAST at HZDR, Germany) with a single-ruler magnetic field been applied. Large eddy simulation (LES) turbulence model is used to resolve the transient details of the melt flow. The predicted time-averaged flow and transient velocity histories are compared with the Ultrasonic Doppler Velocimetry (UDV) measurements and analyzed for both CFD platforms.*
KEYWORDS: EMBR, FLUENT, OPENFOAM, LES, CONTINUOUS CASTING

INTRODUCTION

Continuous casting technology is constantly growing and developing branch of the steel making. With increasing casting speeds and production rates more control is desired for the solidification process to increase quality of the final products. One of the effective technologies to assist the continuous casting (CC) is so called electro-magnetic braking (EMBr). It is applied by inducing an external magnetic field across the CC mold cavity normal to the casting direction to generate Lorentz forces, which slow down the liquid core motion, submeniscus velocities and reduce turbulence level of the hot jets, which are formed due to the fresh melt feeding via submerged entry nozzle (SEN). As it was shown by the authors previously in Ref. [1-3], highly turbulent flow is really undesired due to the remelting of the solidified shell at the hot melt impingment areas; thereby EMBr is a favourable practice for the continous casting process.

Since pioneering works of Takeuchi [4] in the field of numerical simulation of the EMBr process a wide variety of the numerical models appeared at the software market. The requests to the numerical simulation approach have grown over the last decades especially in the field of metallurgical applications as reported by Thomas in Ref. [5]. The computational fluid dynamics (CFD) codes are desired to be robust and effective. Thereby high-performance computing (HPC) technique is widely applied in the modern CFD field.

Nowadays a strong completion is observed between commercial and open-source / in-house codes: the first are typically developed by big professional teams and include wide variety of the numerical models; the later became over the last years a great alternative to the commercial packages, however requiring deep programming and numerical modelling knowledge from the users if it concerns solving complex multiphase problems. On the other hand, the huge advantage of the open-source packages is availability of their programming code for free and possibility to extend and develop them especially for cross-disciplinary tasks.

Based on the long time experience of the authors both with commercial package ANSYS Fluent and open-source CFD software OpenFOAM, the presented study aims to compare their capability to simulate EMBr process “out-of-the-box”. A verification is done based both on experimental and numerical results reported elsewhere in Ref. [6,7, 8].

EXPERIMENTAL SETUP

The aim of the current study is to verify two CFD packages namely ANSYS Fluent and OpenFOAM against the liquid metal experiment excluding and employing electro-magnetic brake [6] as well as against other numerical simulation of turbulent and MHD flows performed by other researches [7, 8]. In the liquid metal experiment GaInSn alloy is used, which is at liquid state at the room temperatures. The details of experiment can be found in corresponding references. The liquid metal properties of the Ga68In20Sn12 alloy used in the experiment are reported by Plevachuk [9].

NUMERICAL MODEL

In the presented studies a standard magnetohydrodynamics module of the ANSYS Fluent commercial CFD software was used to verify an in-house finite volume method (FVM) solver based on the open-source OpenFOAM CFD software package [10]. A new solver was developed by the authors combining arbitrarily incompressible turbulent model and electric potential method to calculated induced current values and MHD forces acting in the fluid.

Basic equations

In the current work an incompressible fluid is considered to represent GaInSn alloy used in the physical experiment. Thereby a turbulent flow including magneto-hydrodynamic effects can be described as a set of Navier-Stokes equations with incompressibility assumption accepted. They are the mass and the momentum conservation equations correspondingly:

$$\nabla \bullet \vec{u} = 0, \tag{1}$$

$$\frac{\partial \vec{u}}{\partial t} + \nabla \bullet (\vec{u} \otimes \vec{u}) = -\frac{1}{\rho}\vec{\nabla} p + \nabla \bullet \boldsymbol{\tau}_{\mathrm{lam}} - \nabla \bullet \boldsymbol{\tau}_{\mathrm{SGS}} + \frac{1}{\rho}\vec{j} \times \vec{B}_0, \tag{2}$$

with velocity $\vec{u}$, liquid density ρ, laminar kinematic viscosity η and pressure p characterizing the fluid flow.

Laminar viscous stress $\boldsymbol{\tau}_{\mathrm{lam}}$ is assumed to be proportional to the symmetric part of the velocity gradient:

$$\boldsymbol{\tau}_{\mathrm{lam}} = 2\eta\, \mathbf{D}, \tag{3}$$

$$\mathbf{D} = \mathrm{symm}(\nabla \vec{u}) = \frac{1}{2}\left(\nabla \vec{u} + (\nabla \vec{u})^{\mathrm{T}}\right). \tag{4}$$

Tensor $\boldsymbol{\tau}_{SGS}$ is the traceless sub-grid scale (SGS) stress tensor, which is evaluated using a turbulence model. Later is discussed in the corresponding section.

To include the influence of the magnetic field the Lorentz force is included into momentum Eq. (2) as a cross product of the current density $\vec{j}$ and the applied magnetic field $\vec{B}_0$. To simulate Lorentz force the electric potential method is applied [11], which is valid at low magnetic Reynolds numbers ($\mathrm{Re_m} \ll 1$): if the induced magnetic field is very small in comparison to the imposed one and does not interfere, it can be neglected. That is mostly the case for the continuous casting process and can be utilized in the presented study. Based on the Faraday’s law

$$\nabla \times \vec{E} = -\frac{\partial \vec{B}}{\partial t} \xrightarrow{\text{with } \vec{B}=\vec{B}_0} \nabla \times \vec{E} \equiv \vec{0} \tag{5}$$

the electric field $\vec{E}$ becomes curl-free due to the constant magnetic field assumption and it can be rewritten using electric potential φ as $\vec{E} = -\vec{\nabla}\varphi$. Thus the electric current is given by the Ohm’s law in a form

$$\vec{j} = \sigma\left(-\vec{\nabla}\varphi + \vec{u} \times \vec{B}_0\right), \tag{6}$$

where σ is the electrical conductivity.

The electric potential φ is derived by solving corresponding Poisson equation derived from conservation equation of the current density $\nabla \bullet \vec{j} = 0$ and taking divergence of the left and right sides of Ohm's law (6):

$$\nabla \bullet \vec{\nabla}\varphi = \nabla \bullet \left(\vec{u} \times \vec{B}_0\right). \tag{7}$$

Electric conductivity σ is considered here to be constant.

Turbulence modelling

The large eddy simulation (LES) based on the sub-grid (SGS) models are successfully applied to the turbulent MHD flows as discussed by Kabayashi [12] and approved in consequent studies to simulate single and multiphase flows been presented elsewhere [7, 8, 13, 14]. Their basic job is to estimate the SGS stress tensor $\boldsymbol{\tau}_{\mathrm{SGS}}$ in a form

$$\boldsymbol{\tau}_{\mathrm{SGS}} = -2C\Delta^2|\mathbf{D}|\mathbf{D}, \tag{8}$$

$$|\mathbf{D}| = \sqrt{2\mathbf{D}:\mathbf{D}}, \tag{9}$$

where C is a SGS model constant.

For the comparison of both CFD software, the standard Smagorinsky (SM) turbulence model [15] is used in the presented study with a Δ-filter of the volume cubic root and $C = C_{\mathrm{S}}^2$.

In the OpenFOAM package two constants C_ϵ and C_{K} are used to define C for the SM SGS model. The default settings $C_\epsilon = 1.048$ and $C_{\mathrm{K}} = 0.094$ are selected, which correspond to C_S used in the ANSYS Fluent package. From the source code analysis the constants relate as

$$(C_{\mathrm{S}})^2 = C_{\mathrm{K}}\sqrt{C_\epsilon / C_{\mathrm{K}}} \tag{10}$$

which gives a value of Fluent SM SGS model parameter $C_{\mathrm{S}} = 0.168$.

Among popular LES models, the wall-adapting local eddy-viscosity (WALE) model is found to be more reasonable and accurate for the complex geometry flows [16]. It resolves the eddy viscosity with the cube of distance close to the wall and does not relay on expensive and complex algorithms or Van-driest damping based on y+ values. The WALE SGS model is used in the current work for the additional set of the OpenFOAM simulations. Since Chaudhary et al. showed in Ref. [7] that standard SM setup is far from experimental measurements, the WALE SGS is employed for the comparative reasons giving actually improved simulation results as it is presented later.

A standard setup both for SM and WALE SGS models is used, thereby the details of the mathematical models are not discussed here and can be found in the corresponding references [7, 12, 15, 16]

SUMULATION RESULTS

General simulation setup

As shown in Fig.1a and referring the GaInSn experiment performed at the HZDR Center [6], the CAD drawing of the simulation domain is prepared using SALOME open-source software. For both CFD software used in the presented work the same mesh produced by the OpenFOAM meshing tool snappyHexMesh is used representing the hex-dominant numerical grid with initially uniform cells size distribution, which are slightly distorted at the regions of the high surface curvature. The details of the numerical grid are presented in Fig.1b; the total number of the finite volumes in the performed study is ~2.7 million cells; no local refinement was applied to avoid well known jumps in the SGS turbulent viscosity for the standard SM. The mesh studies were performed and the size of the mesh was found reasonable both for the accuracy and for the computational performance of the simulations.

Since the LES simulation is always computational costly, all numerical simulations are done for the characteristic time of the simulated process, which can be estimated as

$$t_{\text{charact}} = {}^{\text{V}}\!/_{u_{\text{cast}} \cdot \text{A}_{\text{slab}}}, \tag{11}$$

where V stands for the volume of the simulated domain; $u_{\text{cast}} = 1.35\ \text{m/min}$ is the casting speed ; A_{slab} represents a slab cross-section area. For the presented studies the characteristic time is $t_{\text{charact}} \approx 13\ \text{sec}$.

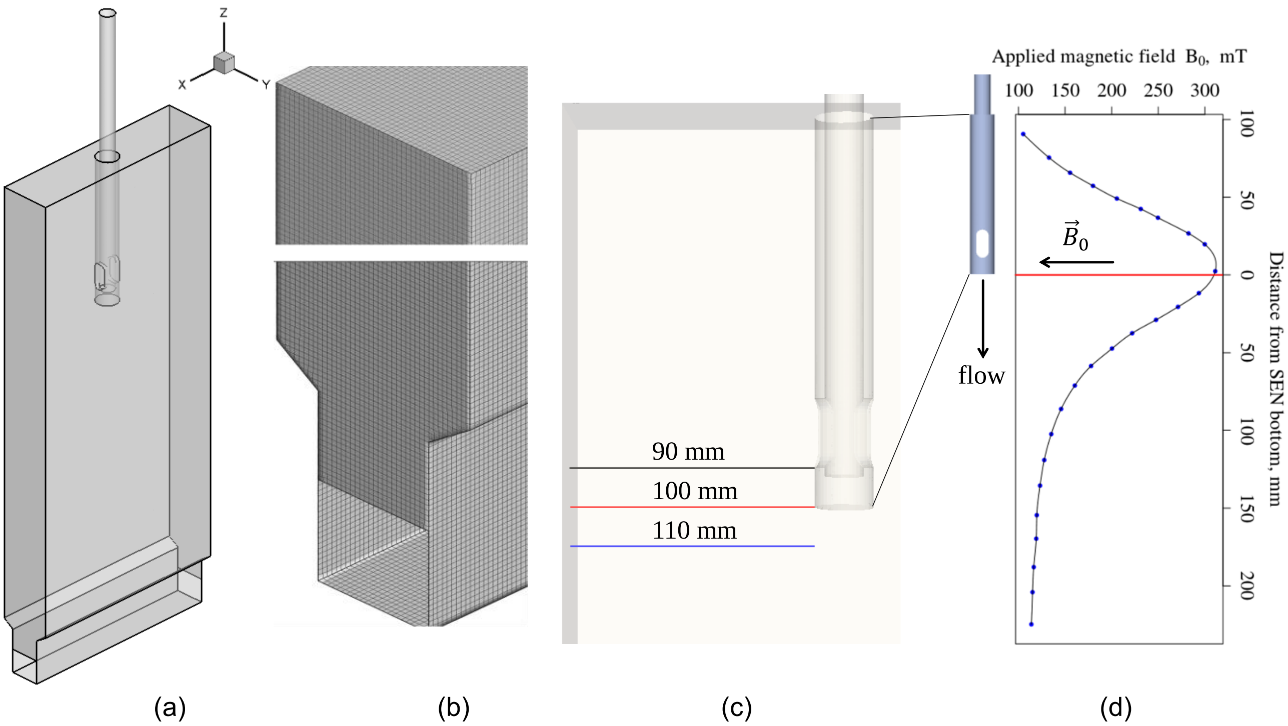


Fig.1 – Simulation domain: (a) LIMMCAST geometry details, see Ref. [8]; (b) numerical grid; (c) monitor lines as given by Chaudhary [7] and Thomas [8]; (d) applied magnetic field normal to the molds wide face (corresponds to case of EMBr at 92 mm in Thomas et al. [8]).

The applied magnetic field $\vec{B}_0$ distribution along casting direction of the mold can be seen in Fig.1d. Its maximum value corresponds to the SEN ports outlet position and reaches its maximum value of ~300 mT.

ANSYS Fluent and OpenFOAM simulation settings

The large eddy simulation with the Smagorinsky SGS model was employed to calculate a transient turbulent flow as described in the previous section for both CFD packages. The external magnetic field considered in this model has a nonzero component in the direction perpendicular to the wide mold face. If consider a higher electrical conductivity of the solidified shell compared to the molten metal in the real continuous casting process, the solidified shell can act as an electrically conducting wall. Thus in the presented work electrically conducting walls are assumed along with the insulating free surface and SEN walls.

INDEX

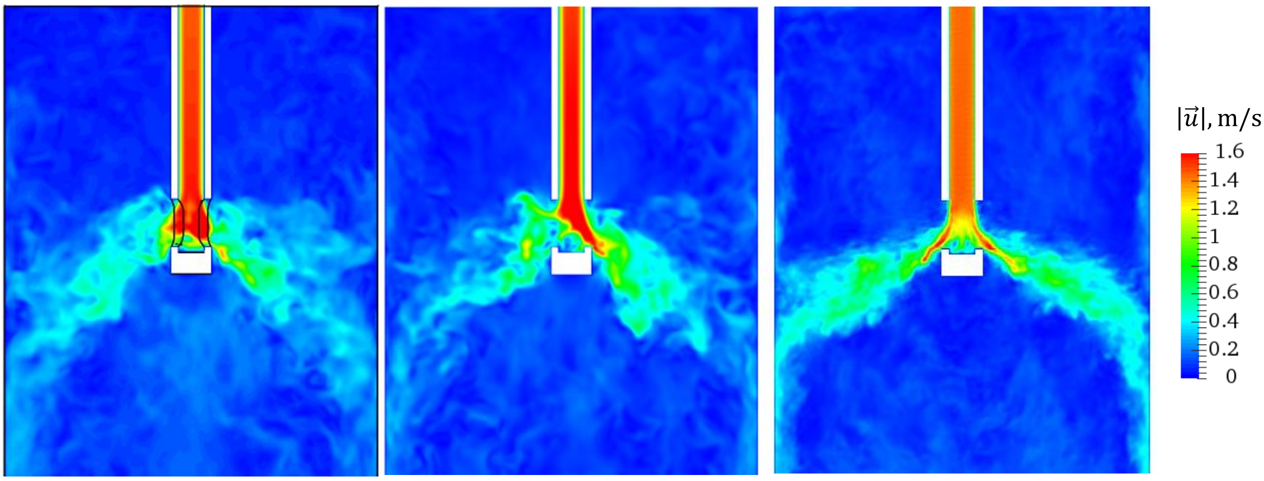
Tab. 1 - Numerical methods, schemes and boundary conditions in ANSYS FLUENT and OpenFOAM simulation

ANSYS Fluent		OpenFOAM	t
Numerical methods		**Case A**	**Case B**
Solver	Pressure-based		
Pressure-velocity coupling	PISO	PIMPLE (blending of PISO and SIMPLE algorithms for transient calculations)	
Spatial discretization			
Pressure	PRESTO	Collocated, Rhie-Chow interpolation [17, 18]	
Momentum	Bounded central differencing	Gauss linear (second order, unbounded)	
Gradient	Least squares cell based	Gauss linear grad(p) leastSquares	
Transient formulation	Bounded second order implicit	backward (transient, second order implicit, potentially unbounded)	
Boundary conditions			
Bulk velocity at nozzle inlet	1.4 m/s		
Pressure at outlet	0 Pa Ref. pressure 1 bar	0 Pa	
Top free surface	Free slip		
Mold wall (wide and narrow)	No slip Electrically conducting		
Turbulence model parameters			
LES SGS	Smagorinsky		WALE
Δ-filter	Volume cubic root		

Similar numerical methods, schemes and boundary conditions are utilized in both CFD codes (ANSYS Fluent and OpenFOAM) in order to ensure the comparability of the simulation results. Table 1 summarizes the main settings of CFD packages been used. It should be emphasized that it is not possible to have the exactly same settings, since the formulation of the models in the CFD codes are different. For the model setup in ANSYS Fluent, primarily the recommended default settings are applied. The settings in OpenFOAM are selected using the benefit of the long-term experience of the authors. A constant time step is employed in the simulations to unsure the Courant number criterion to satisfy a condition $\mathrm{Co} < 0.5$.

In order to reduce the calculation time, the turbulent flow simulations are initialized by running a steady state flow simulation using standard k-ε Reynolds-averaged (RANS) turbulence model. After obtaining the required level of the flow convergence, the LES-type simulation was performed for a characteristic time of 13 sec (see Eq. (11)) to obtain statistically averaged properties of the transient flow to preform comparison and further analysis of the numerical and experimental results. Simultaneous time averaging of the results was carried out to extract the mean flow velocity characteristics.

INDEX



(a) (b) (c)

Fig.2. – Results of the turbulent flow simulation, instantaneous velocity magnitude field: (a) ANSYS Fluent; OpenFOAM (b) Case A and (c) Case C.

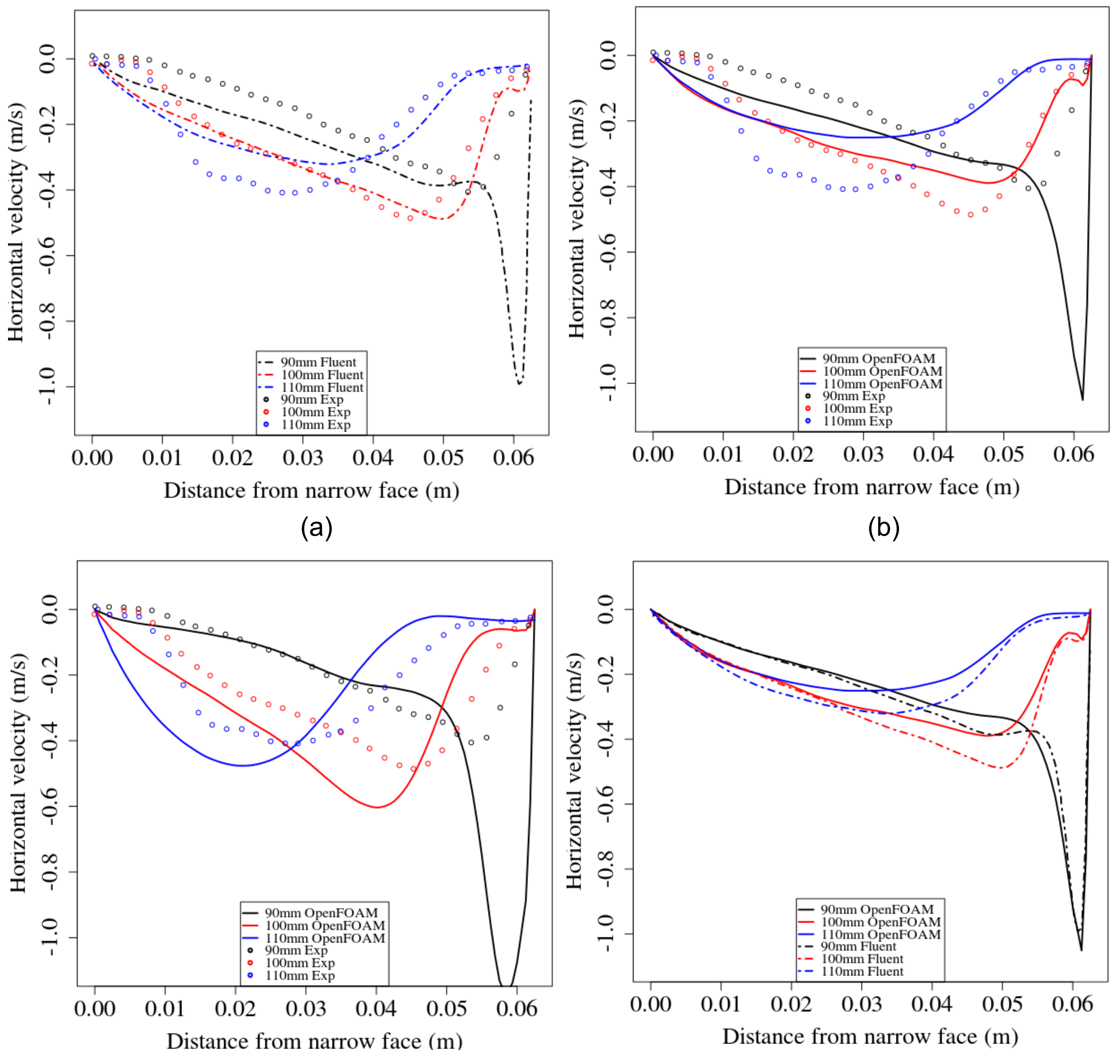


Fig.3. – Time averaged horizontal velocity component u_x along monitor lines marked in Fig.1c; no EMBr applied: (a) ANSYS Fluent vs. experimental data in Ref. [7]; OpenFOAM (b) Case A and (c) Case B vs. experiment al data in Ref. [7]; (d) ANSYS Fluent vs. OpenFOAM Case A.

INDEX

Flow simulation results

The results of the turbulent flow simulations using ANSYS Fluent and OpenFOAM solver are shown in Fig.2. It represents the instantaneous velocity field magnitude in the mid-plane of the casting mold. It can be qualitatively seen that for both software packages using SM SGS model (Fig.2a-b) the flow pattern looks very similar. In Case B for OpenFOAM model employing WALE SGS approach significantly different flow is observed with more stable and developed jets towards the narrow faces of the mold cavity. A very fine eddy structure is detected for the WALE SGS as well in comparison to the standard Smagorinsky LES results.

To perform a quantitative comparison with the measurements at the LIMMCAST experiment in HZDR lab [6] are used, which are also reported by Chaudhary and coauthors in Ref. [7]. Referring to Fig.3 one can see time averaged horizontal component u_x of the velocity vector along monitoring lines at 90 mm, 100 mm and 110 mm below the free surface, which were reconstructed in the experiment based on the UDV method. Both CFD packages could succeed in resolving flow magnitudes towards the narrow face along 100mm line. However they are way off in predicting jet's horizontal velocity below the SEN bottom position at 100m (blue lines and dots) underestimating its magnitude at the second half of the mold closer to the narrow face. Both models are quite good for the jet core velocities at the port outlet line (90mm), but fail in the vicinity of the SEN. However it should be mentioned, that even a tuned SGS model reported by Chaudhary et al in Ref. [7] being precise in the jet core and in the bulk have the same troubles calculating close to SEN. Thereby some additional studies are required here.

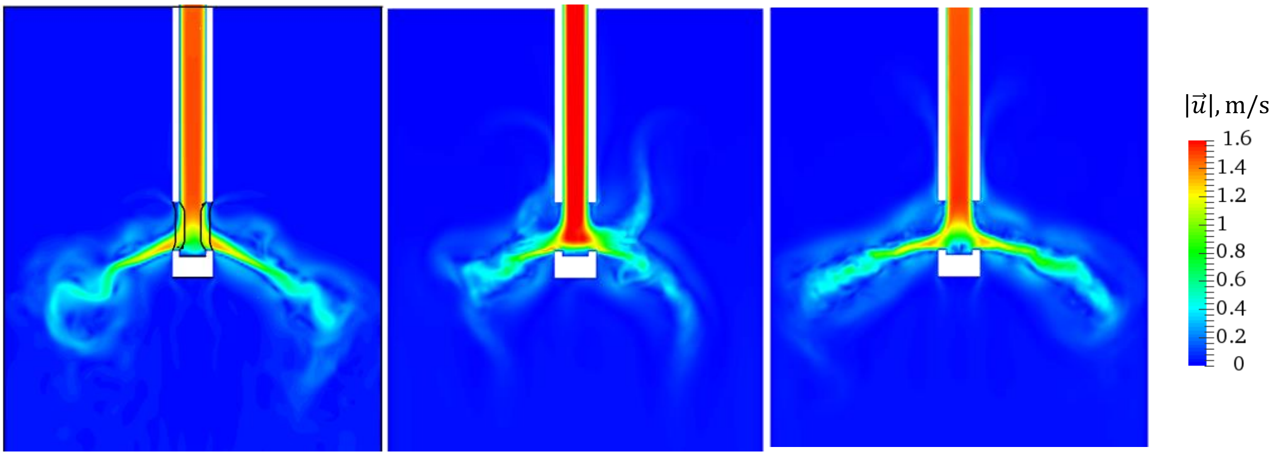
Interesting and promising results (see Fig.3c) are obtained with the WALE SGS model (Case B for the OpenFOAM simulations), which is almost perfect at the jet core (at 90 mm line). It is also better following the experimental tendencies than SM SGS at the 100 mm and 110 mm lines (however with some overshoots); it should be mentioned that the simplest LES filtering was applied. Further model improvement using SGS reported by Kabayashi in Ref. [12] will be made in future work.

Comparison of SM simulations in ANSYS Fluent and OpenFOAM is done in Fig.3d. With the same solver setting the results are pretty close to each other. The difference can be explained by different discretization technique, for example a staggered grid approach is used in Fluent against the collocated in OpenFOAM. The mismatch comes from convective and gradient terms discretization as well, especially when limiting and bounding is applied, which is impossible to clarify in case of commercial software.

EMBr simulation results

Next the electro-magnetic brake process is simulated and compared to the results reported in Ref. [8]. From the velocity field magnitude (see Fig.4) significant decrease in the turbulence is detected due to the Lorentz force acting and not just breaking the flow, but also realigning the turbulent structure. The simple Δ-filtering is used in the presented study, which of course cannot fully tackle the development of coherent structures under the magnetic field being applied as it is done by more complex SGS models (see work of Kabayashi and other studies in this field).

The significant mismatch is observed between both simulations using SM (see Fig.5a and 5b). However WALE model gave better agreement with Fluent results and those reported in Ref. [8]. From the time averaged plots of the horizontal velocity u_x in Fig.5 one can see that all models fail in the most measured regions, except of the WALE SGS (Fig.5c) being fine at 110 mm (red line) and perfect at 100 mm (blue line). However the jet's core velocities are correctly calculated by none of the investigated models. Concluding here, an advanced filtering SGS model is required to predict the MHD effects correctly when employing large eddy simulations for EMBr process. Relative comparison of both software results in Fig.5c gave good agreement between CFD packages for SM model taking into account the arguments mentioned in the previous section.



(a) (b) (c)

Fig.4. – Results of the EMBr simulation, instantaneous velocity field: (a) ANSYS Fluent SM SGS; OpenFOAM (a) Case A and (c) Case B.

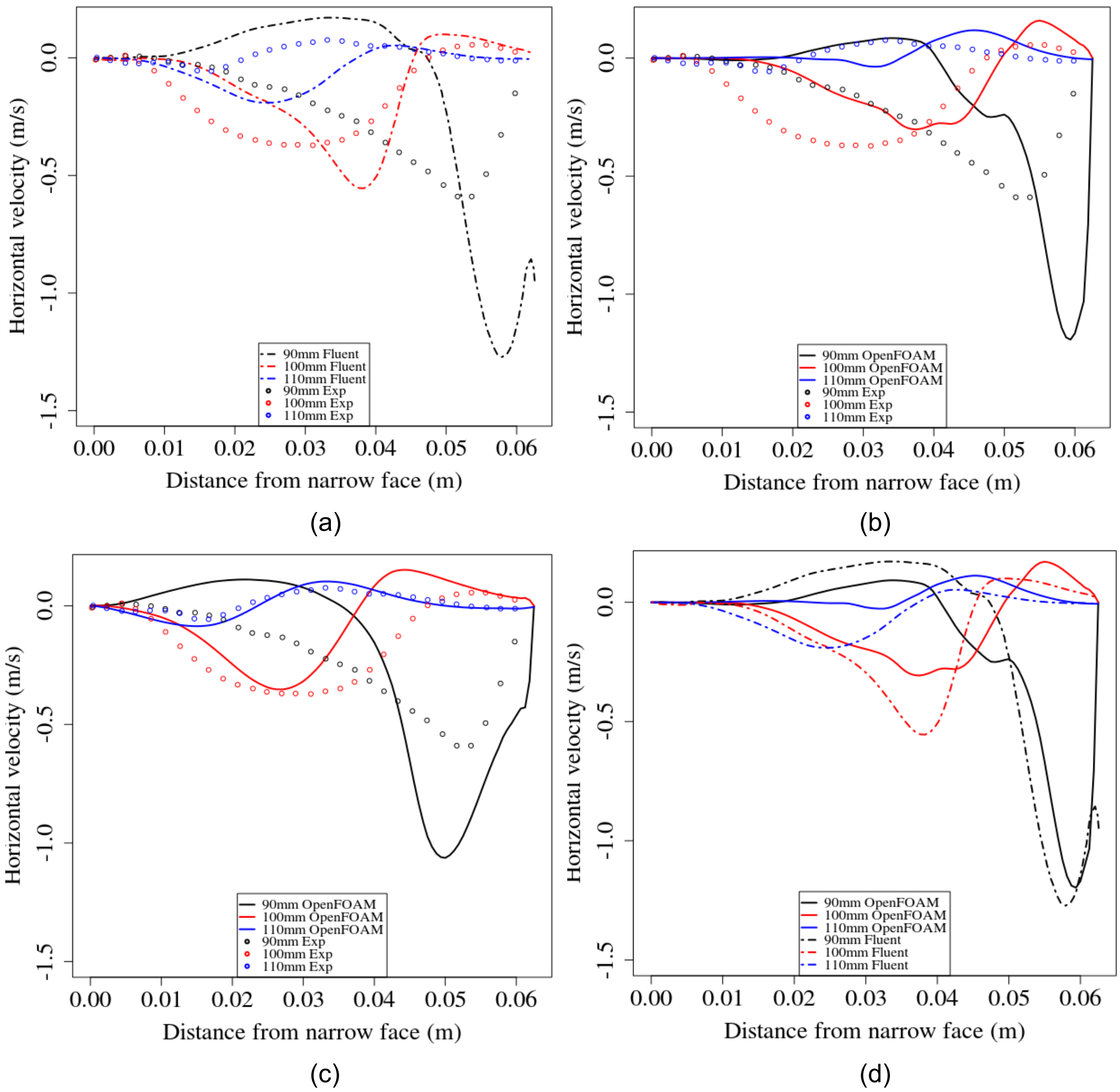


Fig.5. – Time averaged horizontal velocity u_x component along monitor lines (marked in Fig.1c); including EMBr: (a) ANSYS Fluent vs. experimental data in Ref. [8]; OpenFOAM (b) Case A and (c) Case B vs. experimental data in Ref. [8]; (d) ANSYS Fluent vs. OpenFOAM Case A.

INDEX

CONCLUSIONS

The turbulent flow of the GaInSn alloy was investigated in the presented study using both commercial CFD code ANSYS Fluent and in-house solver based on the FVM open-source platform OpenFOAM. The single phase flow with and without magnetic field from the single-ruler EMBr device were simulated and compared with the liquid metal experiment measurements. The comparison studies gave both positive and negative results. In generally it should be concluded that models included in the presented work can predict the turbulent MHD flows with a certain level of inaccuracy due to the fact that they are taken "out-of-the-box" and no model modifications are applied. Using more advanced LES models and improved Δ-filters gives better results; however some necessary improvements are required for the SGS filtering as reported in Ref. [7,8,12].

The difference between results obtained by the authors and those published elsewhere can be explained by following: (a) mesh resolution being used is lower than in reference studies (2.7 M vs. 7.6 M cells); (b) the brass plates, increasing local conductivity of the system and emulating the presence of the solidified shell in the physical experiment, were not included in the simulation; (c) no magnetic effects were considered in the SGS model; (d) the thermo-physical properties of the GaInSn alloy are considered to be constant, despite the report of Plevachuk et al. in Ref. [9] showing their variation due to the temperature, magnetic field etc.; (e) no free surface is considered; (f) general uncertainties related to the discretization practice etc.

ACKNOWLEDGMENTS

The financial support by RHI Magnesita, the Austrian Federal Ministry of Economy, Family and Youth and the National Foundation for Research, Technology and Development is gratefully acknowledged. This work was financially supported by the National Natural Science Foundation of China (No. 51604070). And funding from the China Scholarship Council for study abroad (No. 201706085027).

REFERENCES

1 Wu M, Vakhrushev A, Nunner G, Pfeiler C, Kharicha A, Ludwig A. Importance of Melt Flow in Solidifying Mushy Zone. The Open Transp Phenom J. 2010; 2(1): 16-23. Available from: https://dx.doi.org/10.2174/1877729501002010016

2 Vakhrushev A, Wu M, Ludwig A, Tang Y, Hackl G, Nitzl G. Numerical Investigation of Shell Formation in Thin Slab Casting of Funnel-Type Mold. Metall and Materi Trans B. 2014; 45(3): 1024-1037. Available from: https://dx.doi.org/10.1007/s11663-014-0030-2

3 Wu M, Vakhrushev A, Ludwig A, Kharicha A. . IOP Conf. Ser.: Mater. Sci. Eng. 2016; 117: 012045. Available from: https://dx.doi.org/10.1088/1757-899X/117/1/012045

4 Takeuchi E. Applying MHD technology to the continuous casting of steel slabJOM. 1995; 47 (5): 42–45. Available from: https://dx.doi.org/10.1007/BF03221175

5 Thomas BG. Review on Modeling and Simulation of Continuous Casting. Steel Research Int. 2018; 89: 1700312. Available from: https://doi.org/10.1002/srin.201700312

6 Timmel K, Kratzsch C, Asad A, Schurmann D, Schwarze R, Eckert S. Experimental and numerical modeling of fluid flow processes in continuous casting: results from the LIMMCAST-Project. IOP Conf. Ser.: Mater. Sci. Eng. 2017; 2017 228 012019. Available from: https://dx.doi.org/10.1088/1757-899X/228/1/012019

7 Chaudhary R, Ji C, Thomas BG, Vanka SP. Transient Turbulent Flow in a Liquid-Metal Model of Continuous Casting, Including Comparison of Six Different Methods. Metall and Materi Trans B. 2011; 42: 987-1007. Available from: https://dx.doi.org/10.1007/s11663-011-9526-1

8 Thomas BG, Singh R, Vanka SP, Timmel K, Eckert S, Gerbeth G. J. Manuf. Sci. Prod. 2015; 15 (1): 93-104. Available from: https://dx.doi.org/10.1515/jmsp-2014-0047

9 Plevachuk Y, Sklyarchuk V, Eckert S, Gerbeth G, Novakovic R. Thermophysical properties of the liquid Ga-In-Sn eutectic alloy. J Chem Eng Data. 2014; 59:757–63.

10 Weller HG, Tabor G, Jasak H, Fureby C. A tensorial approach to computational continuum mechanics using object-oriented techniques. Computers in Physics. 1998; 12 (6): 620-631. Available from: https://dx.doi.org/10.1063/1.168744

11 Davidson P. Introduction to Magnetohydrodynamics. Cambridge: Cambridge University Press; 2001.

301-331 p.
12 Kobayashi H. Large eddy simulation of magnetohydrodynamic turbulent channel flows with local subgrid-scale model based on coherent structures. Phys Fluids. 2006;18: 045107. Available from: https://dx.doi.org/10.1063/1.2194967
13 Miao X, Timmel K, Lucas D; Ren Z, Eckert S. Metall and Materi Trans B. 2012; 43: 954-972. Available from: https://doi.org/10.1007/s11663-012-9672-0
14 Liu Z, Li L, Li B. Large Eddy Simulation of Transient Flow and Inclusions Transport in Continuous Casting Mold under Different Electromagnetic Brakes. JOM. 2016; 68: 2180-2190. Available from: https://dx.doi.org/10.1007/s11837-016-1988-9
15 Smagorinsky J. General circulation experiments with the primitive equations. I. The basic experiment. Mon. Weather Rev. 1963; 91: 99-164.
16 Nicoud F, Ducros F. Subgrid-Scale Stress Modelling Based on the Square of the Velocity Gradient Tensor. Flow, Turb. Comb. 1999; 63 (3): 183–200. Available from: https://dx.doi.org/10.1023/A:1009995426001
17 Rhie CM, Chow WL. A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation. AIAA J.1983; 21: 1525–1532.
18 Ferziger JH, Perić M. Computational Methods for Fluid Dynamics. New York: Springer. 2002. Available from: https://dx.doi.org/10.1007/978-3-642-56026-2