



# Numerical Simulation of Multiphase Flow in a Physical Model of a Foundry Degassing Unit

L. Manoch<sup>a, b</sup>, L. Socha<sup>b</sup> , J. Sviželová<sup>b, \*</sup> , K. Gryc<sup>b</sup> , A. Mohamed<sup>b</sup>, J. Häusler<sup>c</sup>

<sup>a</sup>Department of Materials and Engineering Metallurgy, Faculty of Mechanical Engineering, University of West Bohemia, 301 00 Pilsen, Czech Republic

<sup>b</sup>Environmental Research Department, Institute of Technology and Business, 370 01 České Budějovice, Czech Republic

<sup>c</sup>Die-casting Division, MOTOR JIKOV Slévárna a.s., 370 04 České Budějovice, Czech Republic

\* Corresponding author: e-mail: svizelova@mail.vstecb.cz

Received 11.12.24; accepted in revised form 30.01.25; available online 13.06.2025

## Abstract

A physical model of a foundry degassing unit (FDU) uses a simplified approach to melt modelling and degassing. While maintaining the realistic geometry of the rotor, the shape and size of the refining ladle, and the type and amount of inert gas, the original melt is replaced by water. The experimentally evaluated quantity is the degassing efficiency of the melt. Assessing the flow and pressure characteristics in the refining ladle is very complicated experimentally, and it is necessary to use numerical modelling. Numerical modelling of multiphase flow allows the identification, comparison, and quantification of individual flow and pressure characteristics, which can be verified by mutual correlation on basic experimental measurements. Validation of the numerical model is crucial both from the point of view of comparing different states and from the point of view of more advanced multiphase simulations based on this basic model. This article aims to describe the basic numerical model of multiphase flow using the Volume of Fluid (VOF) method for the physical model of the FDU and its verification against experimental measurements.

**Keywords:** Numerical modelling, CFD, Aluminium refining, Multiphase flow, Model validation

## 1. Introduction

The numerical simulation of multiphase flow is an essential tool in various fields of engineering. Particularly in metallurgy, simulations are a source of valuable information that is difficult to detect by experiments during operation due to the unfavourable conditions of high metal temperatures. Simulations, together with operational testing, are also vital in the design and optimisation of aluminium melt refining conditions [1-8]. Furthermore, they provide valuable information on flow dynamics that is unattainable

during operational conditions. Due to the increasing demands for process efficiency and sustainability, accurate and reliable numerical models are necessary to enable the simulation of the behaviour of these complex systems.

The Navier-Stokes equations, which describe the motion of viscous fluids, form the basis for numerical simulations of multiphase flows. These equations are complex and often nonlinear, requiring advanced numerical methods such as finite volume or element methods [9, 10]. In the context of FDUs, the nature of flow and the interactions between fluids, gases and solids are often modelled [11-15]. An equally important aspect is the



definition of the interfaces between the phases, which can be modelled by various approaches, including the front tracking and Volume of Fluid (VOF) methods [16, 17].

Numerical flow modelling involves the use of turbulent models in combination with multiphase models. Turbulent models play a key role in the field of fluid dynamics. The most common group of models for turbulent flow simulation are the so-called RANS (Reynolds Averaged Navier-Stokes) models. These models focus on averaging the Navier-Stokes equations, allowing information to be obtained about the average turbulent flow properties without needing to solve fully turbulent behaviour at the microscopic level [9, 18].

The most widely used RANS models include k-epsilon, k-omega and other hybrid models, characterised by different approaches to modelling turbulent phenomena [18, 19]. Both models have their pros and cons. The k-epsilon model is generally simpler and faster to compute, making it ideal for large-scale simulations. On the other hand, the k-omega model provides better results in regions of high turbulence and near walls, which can be crucial for certain application domains [18, 20].

The k-ε model is one of the most widely used turbulence models, especially in industrial applications. The model was developed by the authors [19, 21] and has become a standard for turbulent flow simulations. This model includes two transport equations: one for turbulent kinetic energy  $k$  (1) and the other for the dissipation of the turbulent kinetic energy  $\varepsilon$  (2). The k-ε model is widely used for simulations in various fields. However, it has limitations, especially in regions with high turbulence or near walls where it may underestimate turbulent flow [18].

$$\frac{\partial k}{\partial t} + U\nabla k = P - \varepsilon + \nabla(v\nabla k) \quad (1)$$

$$\frac{\partial \varepsilon}{\partial t} + U\nabla \varepsilon = \frac{C_1 P - C_2 \varepsilon^2}{k} + \nabla(v\nabla \varepsilon) \quad (2)$$

where  $k$  – turbulent kinetic energy,  $t$  – time,  $U$  – velocity vector,  $P$  – production of turbulence,  $\varepsilon$  – rate of dissipation of turbulent kinetic energy,  $v$  – turbulent viscosity,  $C_1$ ,  $C_2$  – empirical constants.

The k- $\omega$  model has been proposed as an alternative to the k-ε model, with the aim of improving the accuracy of simulations in regions with high turbulence and in regions near the wall of the modelled region [18]. This model also includes two transport equations: one for the turbulent kinetic energy  $k$  (3) and the other for the specific rate of dissipation of turbulent kinetic energy  $\omega$  (4).

$$\frac{\partial k}{\partial t} + U\nabla k = P - \beta k\omega + \nabla(v\nabla k) \quad (3)$$

$$\frac{\partial \omega}{\partial t} + U\nabla \omega = \frac{\gamma \omega P - \beta \omega^2}{k} + \nabla(v\nabla \omega) \quad (4)$$

where  $\omega$  – specific rate of dissipation of turbulent kinetic energy,  $\beta$ ,  $\gamma$  – empirical constants.

The Volume of Fluid (VOF) model is one of the most widely used **multiphase models**, especially in situations where loose interfaces between phases need to be modelled. This model was first introduced by [22] and has since become the standard for fluid dynamics analysis in applications where different phases meet. The

VOF model tracks the interfaces between phases using a fractional function  $F$  that indicates the fraction of each phase in a given volume element and takes values from 0 to 1, where 0 represents purely one phase (e.g., air) and 1 represents the other phase (e.g., water). The basic equations for the VOF model are the phase transport equations (5) and the Navier-Stokes motion equations (6) [22, 23].

$$\frac{\partial F}{\partial t} + \nabla(Fu) = 0 \quad (5)$$

$$\frac{\partial u}{\partial t} + (u\nabla)u = -\frac{1}{\rho}\nabla p + \nu\nabla^2 u + g \quad (6)$$

where  $F$  – Fraction function,  $p$  – pressure,  $\rho$  – density,  $g$  – gravity acceleration.

A key step in the process of fluid flow simulation and analysis is the validation of the numerical model. The primary goal of the validation is to verify that the model correctly reproduces the behaviour of the system it is trying to simulate. This process involves comparing the results obtained from the numerical model with experimental data or analytical solutions to verify the accuracy and reliability of the model. Validation is necessary to ensure that the numerical model correctly captures the physical processes that take place. Inaccuracies in models can lead to erroneous predictions [9].

The aim of this article is to present the procedure for the selection and validation of a numerical model for the CFD analysis of flow in the aluminium refining ladle of the FDU. The numerical simulations, together with physical modelling and operational tests, were carried out as part of the research and development of aluminium melt refining at MOTOR JIKOV Slévárna a.s. The numerical simulations follow on from the research on the efficiency of aluminium refining using a model, with the aims of deepening the knowledge of the real behaviour of the process and proposing measures to increase degassing efficiency.

## 2. Experimental measurement

Continuous measurement of quantities such as impurity concentration or the melt velocity field in the ladle is almost impossible to perform on an FDU under normal operation. Therefore, a physical model of an FDU was used, a more detailed specification of which is provided in reference [24]. It is a water-based physical model where the aluminium melt is replaced with water while argon is used as the refining gas. The model was built to a scale of 1:1, with its dimensions corresponding to the real equipment used in MOTOR JIKOV Slévárna a.s. The experimental measurement was designed for the simplest possible system, i.e. two-phase flow with a clearly defined interface of the two phases – water/air. For this system, the velocity profile and free surface shape can be numerically verified. A refining system for low-pressure casting of aluminium alloys was chosen, characterised by a smaller-size ladle (see Figure 1). The physical model conditions for the measurements are shown in Table 1.

Particle Image Velocimetry (PIV) is commonly used to measure the size and shape of velocity fields. This method uses a laser beam projected through a lens into an area across which the

changes in particle motion are imaged by a high-speed camera. Due to the numerous air bubbles that are created by the spinning rotor causing reflections during the measurement, it was impossible to use this method for velocity field measurements. Instead, a Prandtl probe was used, which was placed at a constant radius of  $R = 0.195$  m with respect to the radius of the refining ladle and was traversed in a direction parallel to the axis of rotation. A diagram of the measurements is shown in Figure 2. The measurement was carried out in 22 positions along the ladle height to obtain a sufficiently accurate velocity profile. The distance between the individual measurement positions was 22 mm. The probe was positioned parallel to the incoming flow, with the probe's directional sensitivity being  $\pm 20^\circ$ . The orientation of the probe during the measurement can be seen in Figure 3. This invasive method was used to obtain the time-averaged velocity at a given location. By plotting the data from each measured point, the time-averaged shape of the velocity profile can be obtained, which can be further used for comparison with the results from the numerical simulation.

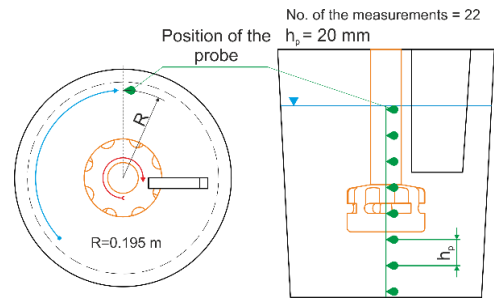


Fig. 2. Diagram of Prandtl probe measurement

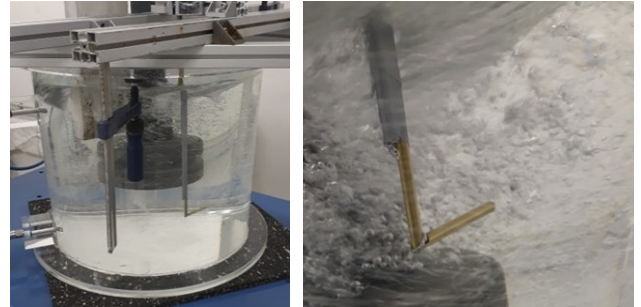


Fig. 3. A view of the Prandtl probe during measurement

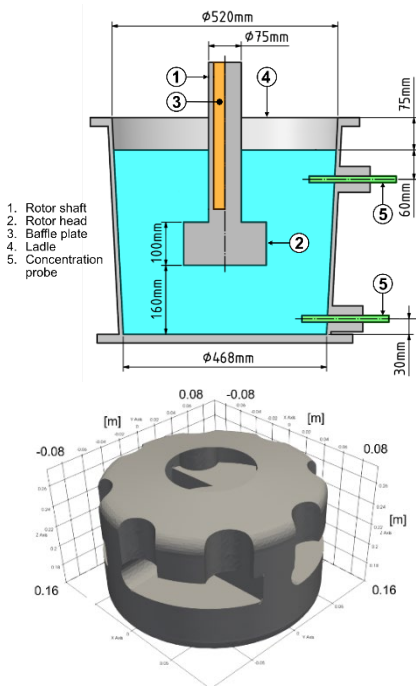


Fig. 1. Dimensional diagram of FDU refining system for LPDC

Table 1. Physical modelling parameters

Parameter	Value	Unit
Number of rotations	350	rpm
Rotor working height	0.16	m
Argon flow	0	$l \cdot \text{min}^{-1}$
Ladle level height	0.43	m

Another value that can be evaluated in experimental measurements and compared with the numerical simulation is the shape of the free surface at the water-air interface. For this purpose, the perimeter of the refining basin was divided into several sections on the top of the refining basin as shown in Figure 4. The value of the free surface resting state and the value of the steady state at constant rotor speed were measured for a specific value of the radius from the axis of rotation. The measured value was the normal distance from the top of the refining basin to free surface. After subtracting the resting and steady-state values, the magnitude of change in the free surface elevation is obtained. Each measurement was taken three times and as a result the average value of the measurements was taken. The change values can be directly compared with the results from the numerical simulation. Argon wasn't used for this kind of experimental measurement.

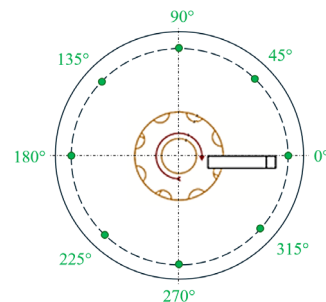


Fig. 4. Diagram of free surface elevation measurement

### 3. Numerical simulations

The first phase of the numerical simulations involved testing the available simulation packages and, based on the agreement with the experimental measurements, selecting the most suitable models for the numerical simulations of aluminium melt refining on the FDU. The software tested was COMSOL Multiphysics 5.6, based on the finite element method and OpenFOAM v2206 based on the finite volume method.

To obtain a solution independent of the computational mesh, a sensitivity analysis of the computational mesh had to be performed for both software programs. Therefore, several meshes of different sizes were created on one geometry. For each of the meshes, integral values were monitored where, unless there was a change or progression of a specific integral value as the size of the computational mesh increased, the solution could be considered independent of the computational mesh. The discretized computational domains are shown in Figure 5.

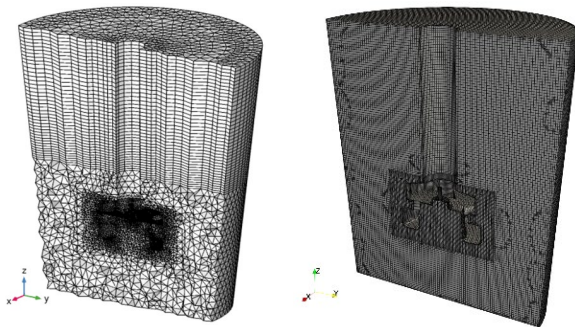


Fig. 5 Examples of computational meshes: COMSOL (left), OpenFOAM (right)

In the case of the studied process, it was a two-phase water-air system. The properties of these media are given in Table 2.

Table 2. The parameters of the model media

Medium	Density ( $\text{kg}\cdot\text{m}^{-3}$ )	Dynamic viscosity ( $\text{Pa}\cdot\text{s}$ )	Temperature ( $^{\circ}\text{C}$ )
Air	1,225	$1,789\cdot 10^{-5}$	20
Water	998,2	$1,003\cdot 10^{-3}$	20

The velocity profile at the Prandtl probe measurement site and free surface behaviour were simulated for both COMSOL and OpenFOAM. For low-pressure systems, in contrast to high-pressure systems, a suitable approach for numerical simulation had to be chosen due to substantial free surface deformation. From the point of view of degassing, the shape of the free surface is not essential, what is important is the total area of free surface for dissipation of the dispersion phase. However, for the validation of the experimental measurement, the shape of the free surface is one of the parameters for comparison and therefore a correct approach to solving it must be chosen. For COMSOL Multiphysics, three different approaches can be chosen to solve the free surface:

- Stationary Free Surface (model  $k-\epsilon$  and  $k-\omega$ )
- Level Set

- Deformed Mesh

The **Stationary Free Surface** method uses the same approach in terms of surface tension as the Deformed Mesh method. The main difference is its use for the stationary solution, where the free surface elevation is solved separately as the last step of the solution based on the pressure distribution. This is the least computationally demanding solution. It is suitable for small deformations and generally applicable as an initial condition for non-stationary solutions. From the viewpoint of large deformations and a purely stationary approach, the Stationary Free Surface method is unsuitable in isolation. It can be used in combination with the Non-stationary Free Surface solution, with the deformation of the network being non-transferable.

The **Level Set** method is a non-stationary method where both phases must be solved. This is the most computationally demanding method and is sensitive to the density of the computational grid, boundary and initial conditions. The Level Set method is unsuitable for production conditions due to its computational complexity, sensitivity to boundary conditions and complicated interfacing with other physical software interfaces.

For the **Deformed Mesh** method, the stiffness of the mesh in the free surface area was additionally addressed to avoid the formation of negative volume elements. This approach is particularly suitable for smaller deformations of the computational mesh. In this approach it is not necessary to solve both phases. Only the main phase (water) was solved and the second phase (air) was represented by the surface tension coefficient  $\sigma$  ( $\text{N}\cdot\text{m}^{-1}$ ). This is a non-stationary solution. In the deformed mesh method, large deformations of the computational mesh occurred, causing the calculation to fail. This problem can be partially eliminated by introducing a step of remeshing the computational mesh based on quality control of the computational mesh elements. However, due to the instability of the calculation, this method was abandoned.

The OpenFOAM software uses the **interFOAM** solver for solving multiphase flow, which uses the VOF (Volume of Fluid) approach, where the fraction of each phase in each cell of the computational mesh is solved. For more advanced simulations also describing diffusion phenomena between the gases, it is necessary to use the Euler-Euler approach, which will be used in future simulations. Two different approaches were also used for the OpenFOAM system. They involved the choice of turbulent models,  $k-\epsilon$  and SST  $k-\omega$ . Identical boundary and initial conditions were used for each of the models. The characteristics of all simulation variants are presented in Table 3.

Table 3. Simulated variants

	Multiphase flow model	Turbulent model	Time dependence	Software
①	Stationary Free Surface	Re $k-\epsilon$	Stationary	COMSOL
②	Stationary Free Surface	SST $k-\omega$	Stationary	COMSOL
③	Level Set	SST $k-\omega$	Non-stationary	COMSOL
④	VOF	Re $k-\epsilon$	Non-stationary	OpenFOAM
⑤	VOF	SST $k-\omega$	Non-stationary	OpenFOAM

## 4. Results and discussion

The velocity profile along the height of the ladle measured by the Prandtl probe is shown in Figure 6. The velocity profile was affected by the presence of the rotor. The highest flow velocity was measured at the point where the fluid stream exits the rotor in the radial direction. This flow divides the volume of the ladle into two circulating areas - the area below the rotor where the flow is more intense and the area above the rotor where the flow is less intense and, especially near the surface, there is a risk of dead volumes [5, 12, 25]. The velocities measured in each of the areas also correspond to this.

The measured free surface elevation values are shown in Figure 7. The values were fairly constant around the perimeter of the ladle and it can be seen that the spinning of the rotor caused a surface elevation of about 0.04 m at the measurement points. The shape of the free surface in the section between 315° and 0° was influenced by the presence of a baffle, which caused a drop in the surface level in the wake behind it.

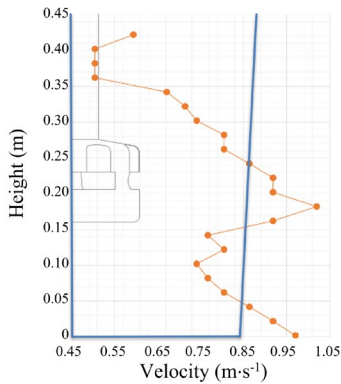


Fig. 6. Velocity profile given by the experiment

In terms of comparing the velocity profiles obtained by the experiment and numerical simulation in COMSOL Multiphysics, we can observe agreement in the velocity profile magnitude, but overall disagreement in the subparts, as seen in Figure 8. In general, a uniform trend of disagreement cannot be determined. The nonstationary solution using the Level Set method (variant ③) shows reasonably good agreement in the first and last thirds of the velocity profile. The middle part of the velocity profile, located in the rotor head region, shows considerable disagreement. In this region, the possible directional insensitivity of the probe must be taken into account.

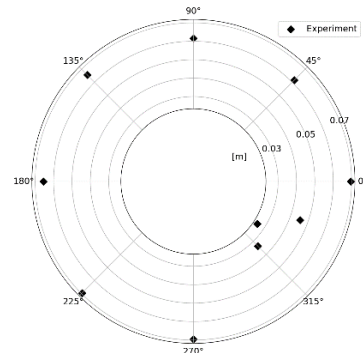


Fig. 7. Free surface elevation given by the experiment

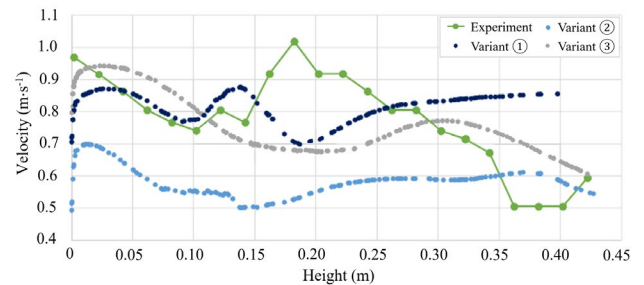


Fig. 8. Comparison of velocity profiles obtained by experiment and numerical simulation in COMSOL Multiphysics

A comparison of the experimental free surface level data with the different approaches in COMSOL Multiphysics is shown in Figure 9. The results indicate good agreement between the experimental data and the Level Set method (variant ③). For the Stationary Free Surface method (variant ① and ②), good agreement can be observed for places without a baffle plate, i.e. for places with constant or relatively small free level elevation. In general, a higher agreement for the Level Set method could be expected. The Stationary Free Surface approach is more suitable for smaller free level deformations. To illustrate, Figure 10 shows the difference between the free level results calculated using the Stationary Free Surface (variant ①) and Level Set (variant ③).

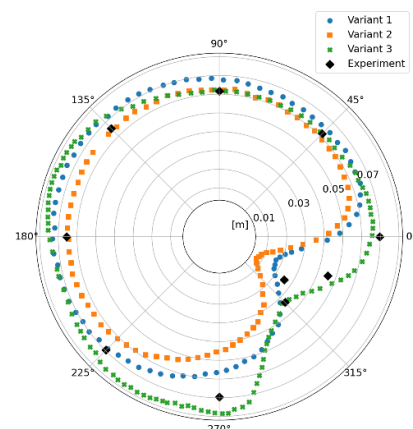


Fig. 9. Comparison of experimental free surface level data with different approaches in COMSOL Multiphysics

For the evaluation of the velocity profile calculated by OpenFOAM, the deflection of the velocity vector at the measurement point was verified using the Prandtl probe. The directional insensitivity zone of the probe was evaluated from the results at the individual time steps of the non-stationary solution. This was done for the X-Y and X-Z planes. The overlay of the insensitivity zone with the measured velocity profile is shown in Figure 11. It can be seen that the limit values for the directional insensitivity of the Prandtl probe can be exceeded for certain points, with the resulting velocity magnitudes subsequently distorted. These are especially regions near the bottom where the directional insensitivity exceeds the critical 20°, and the rotor head region.

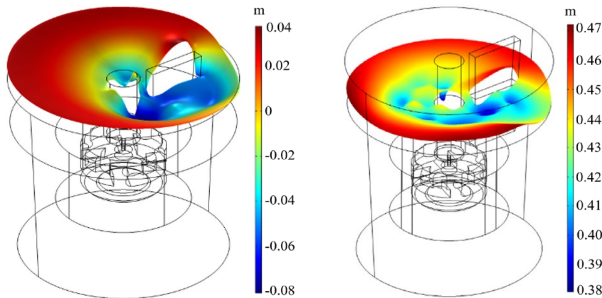


Fig. 10. Illustration of free surface shape of variant ① (left) and variant ③ (right) - COMSOL Multiphysics

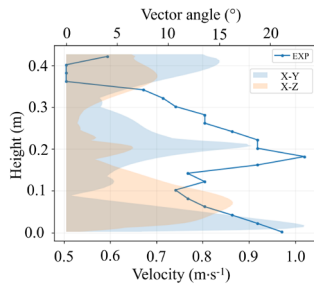


Fig. 11. Position of the probe's directional sensitivity zone

The graph in Figure 12 compares the velocity profile obtained by the experiment and numerical simulation in OpenFOAM software using the Re  $k-\epsilon$  and the SST  $k-\omega$  turbulent models (variant ④ and ⑤). For variant ④, good agreement was not achieved. From a global point of view, the agreement between the variant ⑤ and the experiment is good. The disagreement in the free surface region can be considered to be due to the averaging of the simulation results. The velocity profile from the numerical simulation was generated as an average value over time over three complete rotor revolutions. The insensitivity band in the extreme regions of the velocity profile must also be taken into account.

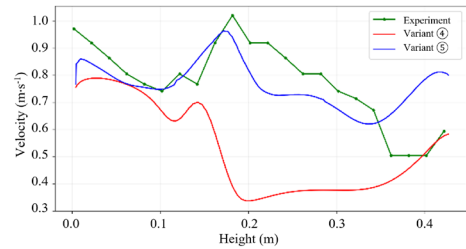


Fig. 12. Comparison of velocity profile obtained by experiment and numerical simulation in OpenFOAM - variant ⑤

Figure 13 compares the free surface elevation obtained by experiment and calculated by numerical simulation in OpenFOAM. Variant ④ showed a relatively large deviation from the experimental measurement results. As can be seen, a good agreement was again achieved for the turbulent SST  $k-\omega$  model and the two-phase VOF model (variant ⑤) in the region both affected and not affected by the baffle plate. The free surface shape of variant ⑤ is shown in Figure 14 for illustration.

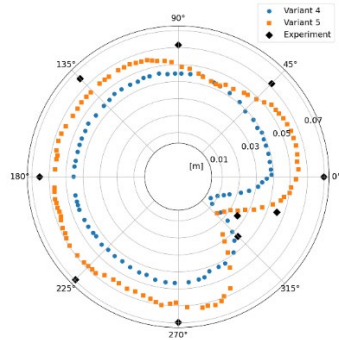


Fig. 13. Comparison of the free surface elevation obtained by experiment with various approaches in OpenFOAM software

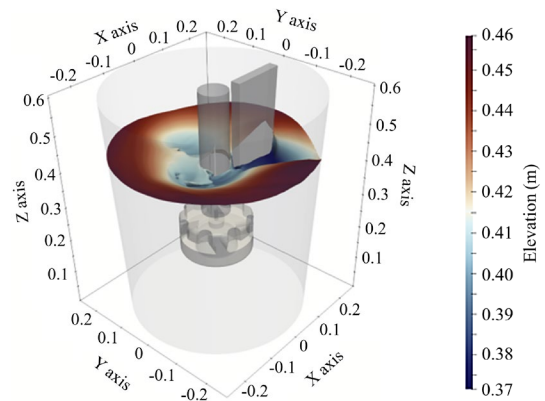


Fig. 14. Example of the shape of the free surface of variant ⑤ - OpenFOAM

## 5. Conclusions

The paper deals with the verification of numerical simulations of flow in an aluminium melt refining ladle on a FDU in COMSOL Multiphysics and OpenFOAM software. Different approaches to modelling multiphase and turbulent flow were tested for this software. These approaches were based on  $k-\epsilon$  and SST  $k-\omega$  turbulent models in combination with different methods for solving two-phase flow, according to the capabilities of each software. Experimental data, measured on a physical model of the FDU, were used for verification. They included a velocity profile measured by a Prandtl probe and free surface elevation.

COMSOL Multiphysics did not achieve very good agreement as far as the velocity profile was concerned. The Level Set method (Variant ③) described the flow in the extreme regions of the velocity profile reasonably well. However, neither of the simulated variants captured the radial flow emanating from the rotor head, causing considerable distortion of the velocity profile.

The velocity profile computed by OpenFOAM using SST  $k-\omega$  (variant ⑤) efficiently reproduced the velocity profile measured by the Prandtl probe. A deviation from the measured values was observed in the region of the liquid level in the ladle. This may have been due to the averaging of the numerical simulation results combined with directional insensitivity of the probe in this region.

The results further suggest that for low pressure systems, the use of COMSOL Multiphysics is limited in terms of the free surface solution. For degassing, it is not necessary to solve the free surface; the possible total area through which the dispersed phase can leave the region and the approximate shape of the free surface in terms of pressure and velocity distribution must be considered. The difference between the areas of the resting free surface and the free surface of the running system may be more than 10%. Good agreement was again achieved for variant ⑤ simulated in OpenFOAM, which solves the free surface with the Volume of Fluid method.

In general, the data match from COMSOL Multiphysics is not as good as from OpenFOAM. OpenFOAM provides more advanced tools for solving complex fluid and gas simulations.

The OpenFOAM system has been chosen for follow-up numerical simulations for the reasons mentioned above, with flow to be solved with the turbulent SST  $k-\omega$  model in combination with the VOF model to calculate the free surface behaviour.

## Acknowledgements

The paper was funded by the Technology Agency of the Czech Republic within the EPSILON programme, as part of the project of Reg. No. TH04010449 "Research and development of refining technologies for increasing of quality of aluminium alloys for high-performance quality castings".

## References

- [1] Saternus, M., Merder, T. & Warzecha, P. (2011). Numerical and physical modelling of aluminium barbotage process. *Solid State Phenomena*. 176, 1-10. DOI: 10.4028/www.scientific.net/SSP.176.1.
- [2] Yamamoto, T., Takahashi, H., Komarov, S.V., Shigemitsu, M., Taniguchi, R. & Ishiwata Y. (2021). Physical modeling of rotary flux injection in an aluminum melting furnace. *Metallurgical and Materials Transactions B*. 52(5), 3363-3372. DOI: 10.1007/s11663-021-02265-9.
- [3] Liu, X., Zhang, Z., Hu, W., Le, Q., Bao, L., Cui, J. & Jiang, J. (2015). Study on hydrogen removal of AZ91 alloys using ultrasonic argon degassing process. *Ultrasonics Sonochemistry*. 26, 73-80. DOI: 10.1016/j.ultsonch.2014.12.015.
- [4] Yamamoto, T., Kato, K., Komarov, S.V., Taniguchi, R. & Ishiwata, Y. (2020) Evaluation of aluminum dross generation rate during mechanical stirring of aluminum through model experiment and numerical simulation. *Metallurgical and Materials Transactions B*. 51(4), 1836-1846. DOI: 10.1007/s11663-020-01842-8.
- [5] Abreu-López, D., Amaro-Villeda, A., Acosta-González, F., González-Rivera, C. & Ramírez-Argáez, M. (2017). Effect of the impeller design on degasification kinetics using the impeller injector technique assisted by mathematical modeling. *Metals*. 7(4), 132, 1-14. ISSN 2075-4701. DOI: 10.3390/met7040132.
- [6] Hernández-Hernández, M., Cruz-Mendez, W., González-Rivera, C. & Ramírez-Argáez, M. A. (2014). Effect of process variables on kinetics and gas consumption in rotor-degassing assisted by physical and mathematical modeling. *Materials and Manufacturing Processes*. 30(2), 216-221. DOI: 10.1080/10426914.2014.952303.
- [7] Mancilla, E., Cruz-Méndez, W., Ramírez-Argáez, M.A., González-Rivera, C. & Ascanio, G. (2019). Experimental measurements of bubble size distributions in a water model and its influence on the aluminum kinetics degassing. *The Canadian Journal of Chemical Engineering*. 97(S1), 1729-1740. DOI: 10.1002/cjce.23432.
- [8] Gyarmati, G., Fegyverneki, G., Tokár, M. & Mende, T. (2021). The effects of rotary degassing treatments on the melt quality of an Al-Si casting alloy. *International Journal of Metalcasting*. 15(1), 141-151. DOI: 10.1007/s40962-020-00428-z.
- [9] Versteeg, H.K. (2007). *An introduction to computational fluid dynamics: the finite volume method*. (2nd ed.). Harlow: Pearson/Prentice Hall.
- [10] Wendt, J.F. (2008). *Computational Fluid Dynamics: An Introduction*. (3rd ed.). Heidelberg: Springer Berlin.
- [11] Abreu-López, D., Dutta, A., Camacho-Martínez, J.L., Trápaga-Martínez, G. & Ramírez-Argáez, M.A. (2018). Mass transfer study of a batch aluminum degassing ladle with multiple designs of rotating impellers. *Journal of the Minerals, Metals, and Materials Society*. 70(12), 2958-2967. DOI: 10.1007/s11837-018-3147-y.
- [12] Yamamoto, T., Suzuki, A., Komarov, S.V. & Ishiwata, Y. (2018). Investigation of impeller design and flow structures in mechanical stirring of molten aluminum. *Journal of Materials Processing Technology*. 261, 164-172. DOI: 10.1016/j.jmatprotec.2018.06.012.
- [13] Gómez, E.R., Zenit, R., Rivera, C.G., Trápaga, G. & Ramírez-Argáez, M.A. (2013). Mathematical modeling of fluid flow in

- a water physical model of an aluminium degassing ladle equipped with an impeller-injector. *Metallurgical and Materials Transactions B*. 44, 423-435. DOI: 10.1007/s11663-012-9774-8
- [14] Warke, V.S., Shankar, S. & Makhlouf, M.M. (2005). Mathematical modeling and computer simulation of molten aluminum cleansing by the rotating impeller degasser. *Journal of Materials Processing Technology*. 168(1), 119-126. DOI: 10.1016/j.jmatprotec.2004.10.016.
- [15] Mirgaux, O., Ablitzer, E., Waz, E. & Bellot, J.P. (2009). Mathematical modeling and computer simulation of molten aluminum purification by flotation in stirred reactor. *Metallurgical and Materials Transactions B*. 40B, 363-375. DOI: s11663-009-9233-3.
- [16] Merder, T., Saternus, M. & Warzecha, P. (2014). Possibilities of 3D model application in the process of aluminium refining in the unit with rotary impeller. *Archives of Metallurgy and Materials*. 59(2), 789-794. DOI: 10.2478/amm-2014-0134.
- [17] Yamamoto, T., Kato, W., Komarov, S.V. & Ishiwata, Y. (2019). Investigation on the surface vortex formation during mechanical stirring with an axial-flow impeller used in an aluminum process. *Metallurgical and Materials Transactions B*. 50(6), 2547-2556. DOI: 10.1007/s11663-019-01681-2.
- [18] Wilcox, D.C. (2006). *Turbulence modeling for CFD*. (3rd ed.). La Canada: DCW industries.
- [19] Launder, B.E. & Spalding, D.B. (1974). The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*. 3(2), 269-289. [https://doi.org/10.1016/0045-7825\(74\)90029-2](https://doi.org/10.1016/0045-7825(74)90029-2).
- [20] Menter, F.R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*. 32(8), 1598-1605. ISSN 0001-1452. DOI: 10.2514/3.12149.
- [21] Launder, B.E., Spalding, D.B. (1972). *Lectures in mathematical Models of turbulence*. London: Academic Press.
- [22] Hirt, C.W & Nichols, B.D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics*. 39(1), 201-225. DOI: 10.1016/0021-9991(81)90145-5.
- [23] Ferziger, J.H., Perić, M. (2012). *Computational Methods for Fluid Dynamics*. (3rd ed.). Heidelberg: Springer Berlin.
- [24] Prášil, T., Socha, L., Gryc, K., Sviželová, J., Saternus, M., Merder, T., Pieprzyca, J. & Gráf, M. (2022). Using physical modelling to optimize the aluminium refining process. *Materials*. 15(20), 7385, 1-12. DOI: 10.3390/ma15207385.
- [25] Mancilla, E., Cruz-Méndez, W., Garduño, I.E., González-Rivera, C., Ramírez-Argáez, M.A. & Ascanio, G. (2017). Comparison of the hydrodynamic performance of rotor-injector devices in a water physical model of an aluminum degassing ladle. *Chemical Engineering Research and Design*. 118, 158-169. DOI: 10.1016/j.cherd.2016.11.031.