Ref. Ares(2025)5198020 - 30/06/2025

Research Programme of the Research Fund for Coal and Steel

Coal / Steel RTD

Project carried out with a financial grant of the

Research Programme of the Research Fund for Coal and Steel

Project Deliverable Report

Issued on	**06.2025**
Technical Group:	**TGA2**

Mapping, Educating, Training, Applying models in continuous CASTing

Project Acronym	**METACAST**
Grant Agreement Number:	**101155952**
Commencement Date:	**01-10-2024**
Completion Date:	**30-04-2026**
Project Deliverable No(s).:	**D3.1**
Project Deliverable Title(s):	**Fundamentals on Modelling**
Dissemination Level:	**Public**
Due Date:	**30.06.2025**
Author(s):	**A. Atf**, C. Gruber. K. Marx, M. Köster, M. De Santis, D. Capobianco, P. Ramirez Lopez, N. Egido Perez, D. Mier

Cover sheet of 2 subsequent pages

TABLE OF CONTENTS

1. INTRODUCTION 4
2. PHYSICS INVOLVED 4
2.1. Fluid Flow 4
2.2. Multiphase Flow 5
2.3. Heat transfer and Phase Change 6
2.4. Inclusion Transport and Clogging 8
3. MODELLING APPROACHES 8
3.1. Governing equations 8
3.2. Turbulence models 9
3.3. Multiphase models 10
3.4. Solidification models 12
4. NUMERICAL TOOLS 13
4.1. CFD platforms 13
4.2. Mesh Generation Strategies 13
4.3. Time Stepping and Convergence Considerations. 15
4.4. Summary 16
5. REFERENCES 18

1. INTRODUCTION

Numerical modelling is a powerful tool used to better understand and improve the continuous casting (CC) process in steelmaking. It helps simulate important physical processes like fluid flow, mixing of different phases (such as gas and liquid), heat transfer, solidification of steel, and movement of small particles (inclusions) [1], [2]. These simulations make it possible to study areas inside the caster that are difficult to measure directly, and they help engineers adjust the process to improve quality, safety, and efficiency [3]. Continuous casting is a complex process with challenges such as strong turbulence, multiphase flow, and fast temperature changes. Numerical models help deal with these challenges and play a key role in making the steel casting process more reliable and effective [4]. This report gives an overview of the main physical phenomena, modelling methods, and tools used in CC simulations.

In this report, the fundamental physical phenomena involved in continuous casting, the corresponding modelling strategies, and the numerical tools required to simulate them effectively are systematically reviewed to provide a comprehensive understanding of numerical modelling in steelmaking applications.

2. PHYSICS INVOLVED

2.1. Fluid Flow

Fluid flow in continuous casting is primarily governed by the turbulent motion of molten steel within the mold, tundish, and submerged entry nozzle (SEN). The flow behaviour is influenced by casting speed, nozzle design, and any applied electromagnetic forces [4]. Understanding the flow field is essential, as it directly impacts surface quality, entrapment of inclusions, and overall heat transfer. Accurate modelling of this turbulent flow requires resolving complex transient structures, including recirculation zones and jet impingement near the meniscus. [5], [6].

An example of typical velocity flow field (a) inside the mold region and (b) at the mold top surface is shown in **Figure 1** [7].

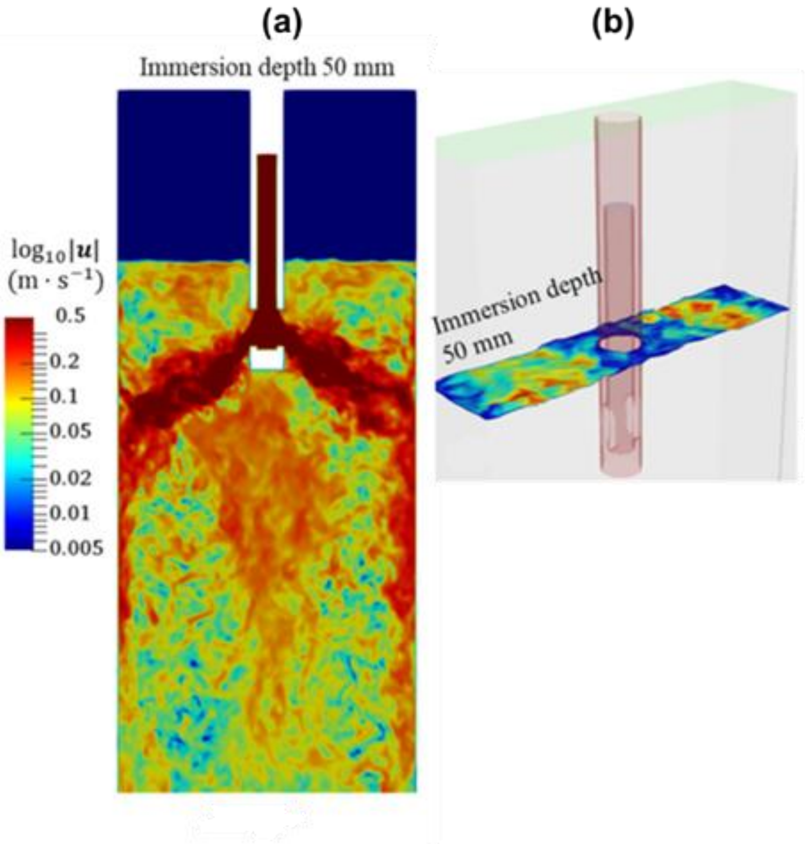

Figure 2 A typical velocity flow field (a)inside the mold region and (b)the mold top surface [7]

2.2. Multiphase Flow

Multiphase flow in CC involves the interaction between the molten steel and injected argon gas, used to prevent clogging in the SEN. The gas phase forms bubbles that influence fluid dynamics, enhance mixing, and may help in inclusion flotation. Capturing the behaviour of dispersed bubbles (coalescence, breakup, and interaction with turbulence) is critical for predicting gas coverage at the meniscus and understanding defect formation mechanisms. Depending on the level of detail required, different multiphase modelling approaches can be used.

The Euler–Euler approach is suitable for dense dispersed flows and is commonly coupled with Population Balance Models (PBM) to represent bubble size distributions. The Volume of Fluid (VOF) method is more appropriate for sharply defined interfaces, such as slag-metal separation. In contrast, the Lagrangian Discrete Phase Model (DPM) tracks individual particles or bubbles and is ideal for dilute phases.

An example of the flow dynamics representing the interaction gas/metal at surface is shown in **Figure 3** [8] . It should be noted that the slag layer is included in the numerical simulation but is absent in the water model, a common practice in water models used for validation. The critical velocity for steel-slag emulsification onset, which influences inclusion entrapment, is largely independent of the presence of a typical slag layer [9]. Therefore, the absence of slag in water models does not significantly affect their validity for studying flow behaviour and gas fraction distribution.

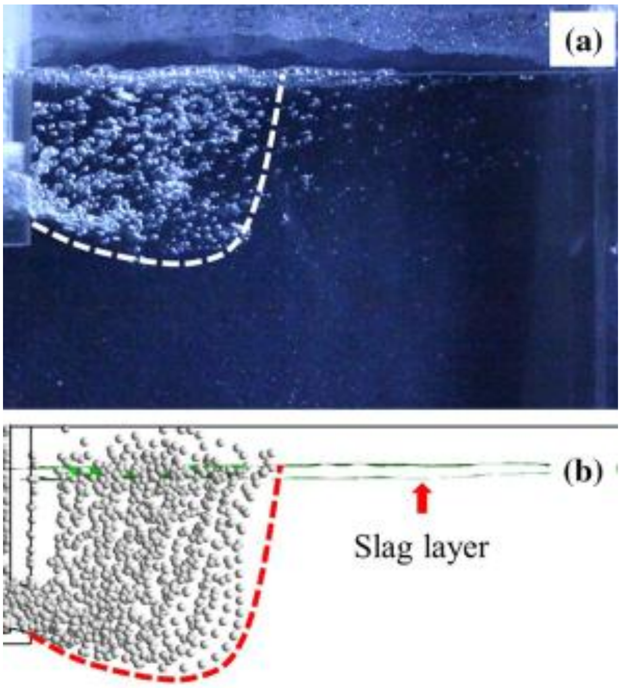

Figure 4 Air-Water Multiphase Flow in SEN and Mold: Experiment vs. CFD simulation [8]

2.3. Heat transfer and Phase Change

The thermal field in continuous casting is strongly coupled with fluid dynamics and the phase change from liquid steel to solid shell. Heat is extracted through the (copper) mold walls and further cooled by secondary spray systems. Accurate modelling of heat transfer involves solving the energy equation while accounting for latent heat during solidification, and properly capturing thermal boundary conditions such as contact resistance, mold flux behaviour, and cooling efficiency. Phase change is typically treated using the enthalpy-porosity method, which allows for a smooth transition between liquid, mushy, and solid zones. Predicting the shell growth rate and its uniformity is crucial for casting stability and quality. [10]

In addition to latent heat, reliable modelling must consider complex thermal boundary conditions such as mold-steel interfacial resistance, mold flux behaviour, and cooling system effectiveness. These factors influence the formation and growth of the solid shell, whose thickness and uniformity are critical for strand stability, breakout prevention, and final product quality.

An example of temperature profile contour in a thin slab caster is shown in **Figure 5** [11].

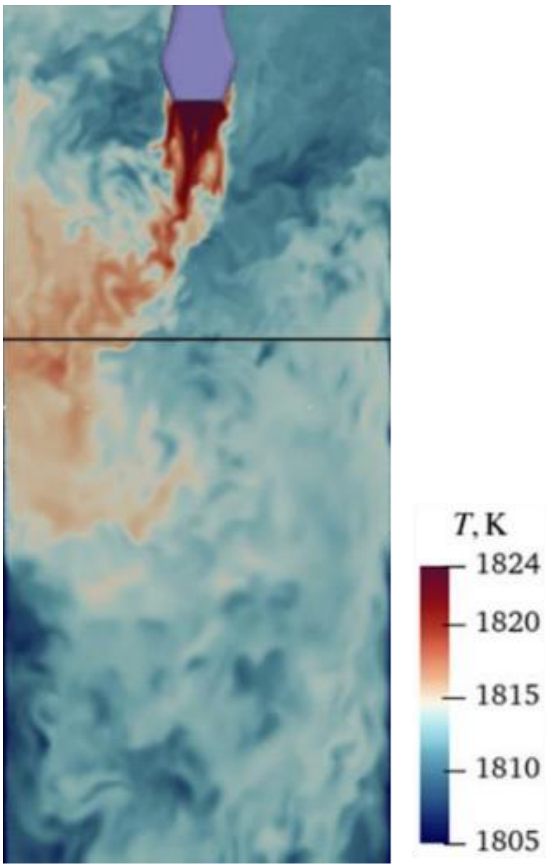

Figure 6 Temperature profile contour in a thin slab caster [11]

2.4. Inclusion Transport and Clogging

Non-metallic inclusions, such as alumina or slag particles, are transported within the molten steel by fluid flow and can lead to product defects if not removed before solidification. Their behaviour is governed by drag, buoyancy, and turbulent dispersion, and can be significantly affected by flow structures, bubble motion, and electromagnetic forces. Inclusion modelling typically involves Lagrangian tracking (via DPM), optionally coupled with models for interaction with bubbles (e.g., attachment and flotation), wall impingement, or entrapment in the solidifying shell. Understanding inclusion trajectories and removal mechanisms is essential for developing clean steel practices and validating quality-focused casting strategies [12]. The following **Figure 7** represents the evolution of the clog in the submerged entry nozzle of an industrial caster in the course of time.

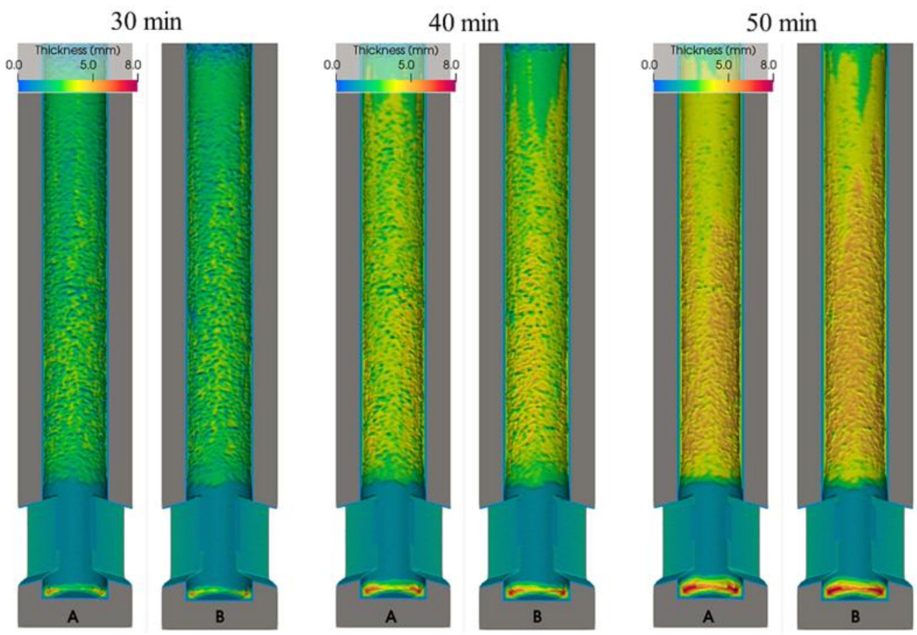

Figure 8 Numerical simulation of the clog growth during time [12]

3. MODELLING APPROACHES

3.1. Governing equations

The core of numerical modelling in continuous casting lies in the governing equations that describe the conservation of mass, momentum, and energy within the system. These equations, derived from the principles of fluid dynamics and thermodynamics, form the basis of Computational Fluid Dynamics (CFD) simulations. The continuity equation ensures mass conservation, while the Navier-Stokes equations govern the momentum transfer, accounting for pressure, viscous, and external forces. The energy equation captures heat transfer through conduction, convection, and phase change, an essential aspect in modelling solidification. In multiphase and reactive systems, additional transport equations may be included to model species concentration, turbulence quantities, or scalar transport (e.g., volume fraction, inclusion concentration). Depending on the modelling approach, such as Eulerian or Lagrangian frameworks, the equations can be modified to suit the physics of interest. These equations are typically solved numerically using discretization techniques (like finite volume or finite element methods), making them suitable for simulation within industrial-scale continuous casting systems.

3.2. Turbulence models

Turbulence plays a critical role in the behaviour of molten steel during continuous casting, particularly in areas such as the mold and tundish, where flow is highly unsteady and chaotic. Direct Numerical Simulation (DNS), which resolves all turbulent scales, is considered the most accurate approach. However, DNS remains impractical for industrial-scale simulations due to its prohibitive computational cost[1], particularly at high Reynolds numbers encountered in steel casting [13]. Instead, Reynolds-Averaged Navier–Stokes (RANS) models are widely employed. Among these, the standard k-ε model is one of the most used for its robustness and computational efficiency, especially for fully developed turbulent flows [14]. It solves two additional transport equations for the turbulent kinetic energy (k) and its dissipation rate (ε). However, it may struggle in cases with strong anisotropy or swirl.

In regions with strong boundary layer development and flow separation, the k-ω model, and particularly its Shear Stress Transport (SST) variant, offers better near-wall resolution and predictive accuracy [15]. It blends the advantages of both k-ε and k-ω models.

[1] Xiaoshu Lü , Tao Lu, Tong Yang, Heidi Salonen, Zhenxue Dai, Peter Droege, and Hongbing Chen. (2021). Improving the Energy Efficiency of Buildings Based on Fluid Dynamics Models: A Critical Review. Energies. 14. 5384. 10.3390/en14175384.

For cases demanding more precise representation of transient and recirculating flows, Large Eddy Simulation (LES) provides improved accuracy by directly resolving large-scale turbulent structures while modelling only the smaller ones. However, LES comes with significantly higher computational requirements. To bridge the gap between accuracy and computational feasibility, hybrid models such as Scale-Adaptive Simulation (SAS) and Detached Eddy Simulation (DES) have been developed. These methods combine features of both RANS (Reynolds-Averaged Navier-Stokes) and LES, making them suitable for complex industrial simulations where full LES remains impractical.

The following **Figure 9** illustrates the differences between three common turbulence modelling approaches applied to a turbulent jet. The RANS model provides a smoothed, time-averaged view of the flow, lacking the fine-scale structures. LES captures the larger, more energetic eddies and reveals detailed transient behaviour. DNS, while offering the highest accuracy by resolving all turbulence scales.

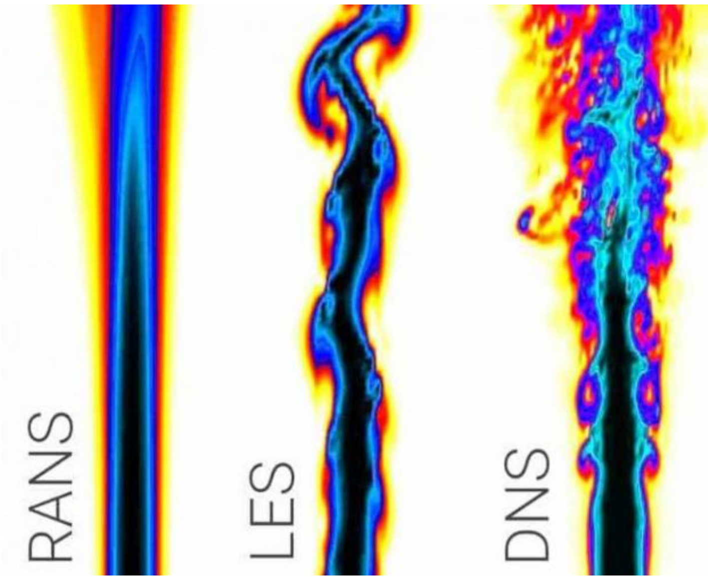

Figure 10 Different approaches for CFD modelling the turbulent jet [16]

3.3. Multiphase models

Multiphase flow is a fundamental aspect of continuous casting, as it involves interactions between molten steel, slag, gas bubbles (e.g., argon injection), and non-metallic inclusions. Capturing these interactions accurately requires appropriate multiphase modelling strategies that reflect the physics of the system, and the scales involved.

The **Volume of Fluid (VOF)** model is used to track the interface between immiscible fluids, such as molten steel and slag. It solves a single set of momentum equations and uses a volume fraction to represent the interface. VOF is especially useful when surface deformation and flow at the interface are important.

The **Euler-Euler model** treats each phase (e.g., gas and liquid) as interpenetrating continua, with separate conservation equations for each. It is suitable for high-volume-fraction multiphase flows, like bubbly flows in the mold or tundish. This model can be extended with Population Balance Models (PBM) to account for bubble size distribution, coalescence, and breakup.

The **Mixture model** is a simplified Eulerian approach that assumes phases move at different velocities but are strongly coupled. It reduces computational cost while still capturing relative motion between phases, making it practical for systems with dispersed gas or particle phases.

The **Discrete Phase Model (DPM)** tracks individual particles or bubbles (example with Argon [17], **Figure 11**) through a continuous fluid field. It is ideal for simulating low-volume-fraction phases like inclusion particles or injected gas bubbles. DPM allows for detailed analysis of particle trajectories, residence time, and interactions with the flow.

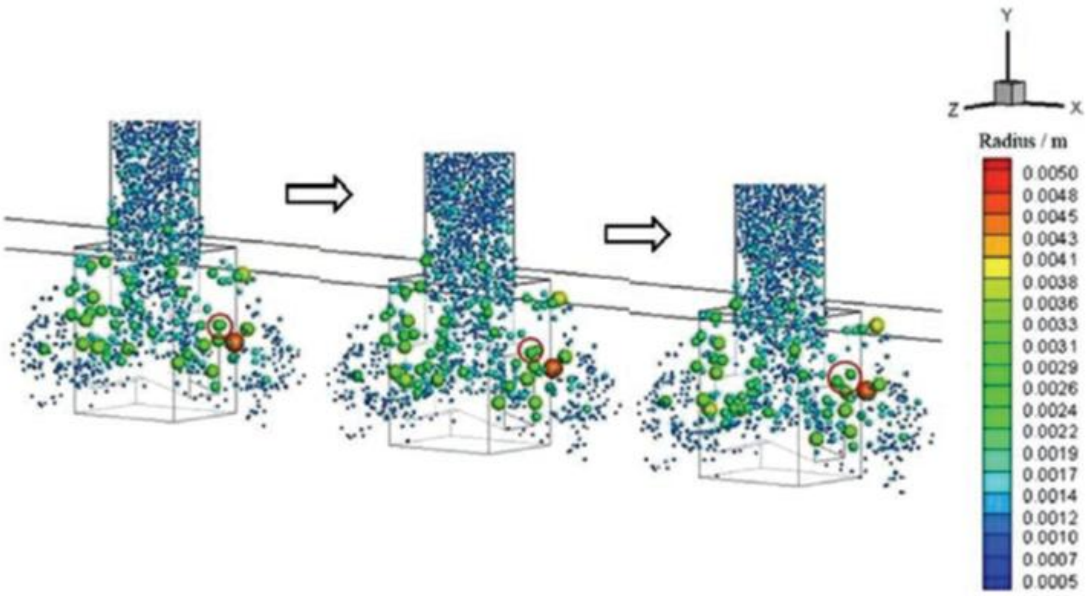

Figure 12 Argon-gas DPM bubbles in the submerged entry nozzle.[17]

Each model offers advantages depending on the type of multiphase interaction being modelled, and their selection should align with the physical characteristics of the system, required accuracy, and available computational resources.

3.4. Solidification models

Solidification is a key process in continuous casting, where the molten steel cools and transforms into a solid shell inside the mold. Modelling this phase change accurately is critical to predict shell thickness, avoid defects, and ensure product quality. The enthalpy-porosity method is widely used to simulate solidification in computational fluid dynamics (CFD). This approach integrates the latent heat of fusion into the energy equation through an enthalpy formulation. It treats the mushy zone (the region where solid and liquid coexist) as a porous medium, where the flow velocity is gradually reduced by introducing a damping term proportional to the solid fraction. This enables smooth and efficient handling of the phase change without explicitly tracking the solid-liquid interface. In the context of continuous casting, where the solidified steel is continuously withdrawn from the domain, the enthalpy-porosity model also accounts for the finite pulling velocity of the solid phase (sketch in **Figure 13** [18]). This motion is incorporated into the momentum equations and helps accurately represent the thermal and flow behaviour near the solidification front.

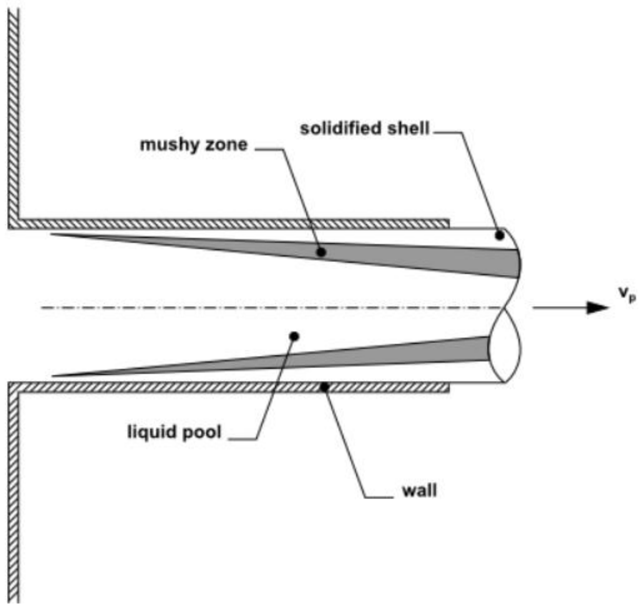

Figure 14 Pulling a solid in continuous casting. [18]

4. NUMERICAL TOOLS

4.1. CFD platforms

A variety of computational fluid dynamics (CFD) software platforms are available for simulating continuous casting processes. ANSYS Fluent is one of the most widely used commercial tools, known for its robust solvers, extensive physical models, and user-friendly interface. OpenFoam, an open-source alternative, offers high flexibility and customization, making it popular in research environments where code modification is required. CFX, also commercial, provides strong Multiphysics capabilities and efficient solvers optimized for heat transfer problems. Each platform has strengths that can be used depending on the complexity of the model, available resources, and specific project needs.

4.2. Mesh Generation Strategies

Mesh generation is a critical step in CFD modelling, directly impacting both the accuracy and numerical stability of simulations. A well-designed mesh ensures that essential geometric features are accurately represented and that numerical errors are minimized.

In continuous casting simulations, structured meshes, composed of regularly arranged elements, are preferred for simple domains like straight channels due to their superior accuracy and computational efficiency. For more complex geometries, unstructured or hybrid meshes are often required to fit complex shapes while keeping the mesh quality high.

To improve solution accuracy, local mesh refinement should be applied in regions with strong gradients of velocity, temperature, or phase fraction, such as the meniscus, jet impingement zone, and solidification front. It is essential to monitor mesh quality indicators such as aspect ratio, orthogonality, and skewness, as poor quality in these metrics can cause convergence issues and compromise results.

Near-wall behaviour should be resolved using inflation layers to accurately capture velocity and thermal boundary layers, which are crucial for predicting heat extraction and surface quality in the mold. Additionally, performing a mesh independence study is considered best practice, ensuring that further mesh refinement does not significantly alter the results and that the solution is robust.

Mesh Topologies: 2D and 3D Element Types

Meshes can be built using a variety of element topologies, depending on the dimensionality and complexity of the geometry. The selection of element type influences solution accuracy, computational cost, and mesh generation time.

2D Mesh Elements:

- Triangles: Highly flexible and easy to generate for complex domains; common in unstructured meshes. However, they may introduce numerical diffusion if not properly aligned with the flow.
- Quadrilaterals: Preferred in structured meshing for their superior numerical performance, especially in boundary layers.

3D Mesh Elements:

- Tetrahedra: Widely used in unstructured meshing due to ease of generation. Less accurate than hexahedra, requiring more elements for similar resolution.
- Hexahedra (Bricks): Ideal for structured meshing. Offer low numerical diffusion and high efficiency, especially in regular domains.
- Wedges (Prisms): Commonly used in boundary layers and transitional regions between tetrahedral and hexahedral zones.
- Pyramids: Employed to connect different element types (e.g., hexahedra to tetrahedra) in hybrid meshes.
- Polyhedra: Offer enhanced accuracy and numerical stability by efficiently representing irregular volumes with fewer elements.

Each mesh type presents trade-offs in terms of complexity, quality, and solver performance. **Figure 15** [18] below illustrates commonly used 2D and 3D mesh element types in CFD simulations, where **Figure 16** [19] shows a sample of a computational domain with hybrid mesh with boundary layer.

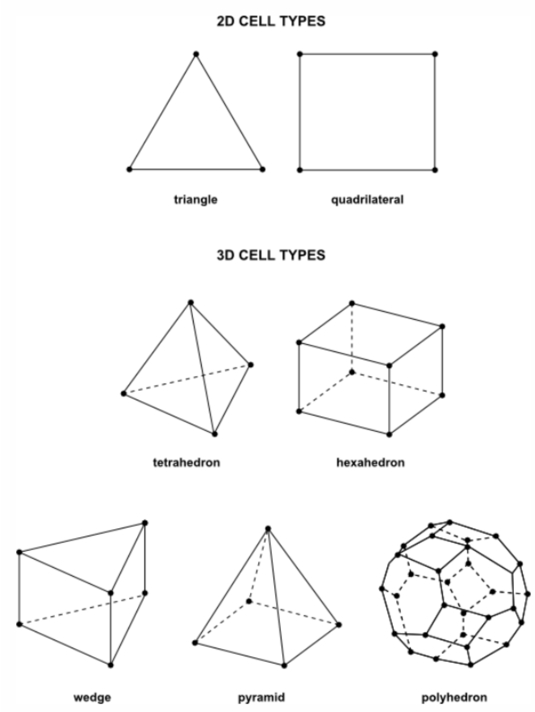

Figure 17 Common 2D and 3D mesh element types used in CFD modelling. [19]

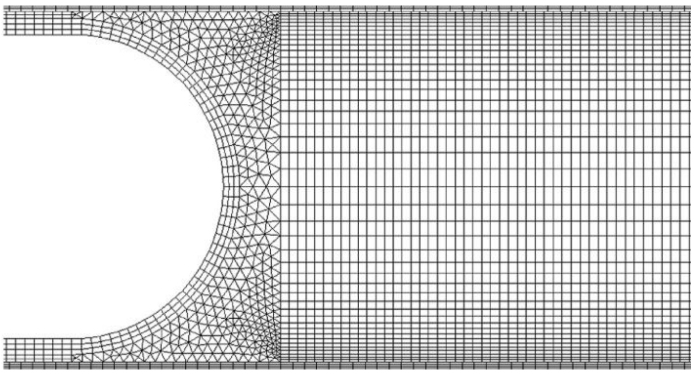

Figure 18 A sample of a computational domain with hybrid mesh with boundary layer. [20]

4.3. Time Stepping and Convergence Considerations

Choosing the right time step is key to capturing the dynamic behaviour of continuous casting processes, such as fluctuating flow, transient multiphase interactions, and the progression of solidification. A good starting point is to estimate the physical time scales of interest (such as how quickly the fluid moves across the domain or how fast solidification progresses) and then choose a time step small enough to resolve these effects without making the simulation unnecessarily long. One of the most practical tools for time step control is the Courant number (Co), which should generally remain below 1 in transient simulations to ensure numerical stability, especially when using explicit schemes. For implicit solvers, a slightly higher Courant number may still yield stable results but keeping it under 1 is a safe guideline when starting out.

Using adaptive time stepping if available (this allows the solver to automatically reduce the time step in regions of rapid change and increase it when the solution stabilizes), improves efficiency. It's also critical to monitor convergence at each time step, not just by checking that residuals fall below a threshold (e.g., 1e-4 or 1e-5), but also by ensuring mass, momentum, and energy balances are satisfied, and that key physical quantities (e.g., velocities, temperatures, phase fraction) behave smoothly over time. Conducting an initial (not long) simulation allows the user to evaluate the characteristic time scales of the system, such as flow development or thermal response, which in turn provides a practical basis for selecting appropriate time step sizes and setting effective convergence criteria for the full simulation.

4.4. Summary

This report presents a fundamental overview of numerical modelling in continuous casting, emphasizing its role in improving process efficiency, product quality, and operational stability. By simulating key phenomena, such as turbulent fluid flow, multiphase interactions, heat transfer, solidification, and inclusion transport, CFD provides critical insight into the inaccessible regions of the caster.

Turbulence is modelled using approaches ranging from the standard k-ε and k-ω models to more advanced LES and SAS methods, balancing accuracy with computational cost. Multiphase flows are captured through VOF, Euler-Euler, and DPM methods, enabling prediction of bubble dynamics and inclusion behaviour. Solidification modelling uses energy-based methods like the enthalpy-porosity approach to simulate shell growth and phase transitions.

Mesh generation, time stepping, and solver selection are also key to achieving accurate results. High-quality meshes with proper refinement and boundary layer resolution, along with convergence and mesh-independence checks, are essential.

The table below summarizes the key numerical modelling aspects in continuous casting, outlining the commonly used methods, their practical applications, strengths, and associated challenges, providing a brief overview of the current state of the art.

Table1 Summary of State-of-the-art

Modelling Aspect	Common Methods	Applications in Continuous Casting	Advantages	Challenges/Limitations
Turbulence Modelling	Standard k-ε k-ω LES SAS RANS	Capturing complex flow patterns and turbulence effects in mold and tundish	Good compromise between accuracy and cost (RANS), high accuracy (LES)	LES computationally expensive, RANS may lack detail
Multiphase Flow	VOF, Euler-Euler, Mixture Model, DPM	Modelling gas-liquid flows, slag behaviour, inclusion transport	Accurate interface tracking (VOF), flexible phase interaction (Euler-Euler)	High computational demand, complex coupling
Solidification Modelling	Enthalpy-Porosity Method, Mushy Zone Models	Simulating phase change, shell growth, heat transfer	Efficient handling of phase change in CFD	Requires calibration, mushy zone complexity
Numerical Tools	ANSYS Fluent, OpenFoam, CFX	CFD platform selection depending on problem scale and resources	Adaptable, validated solvers, user support	Licensing costs (commercial tools)
Mesh generation	Structured, Unstructured, Hybrid	Ensuring accurate geometry representation and solver stability	Adaptive meshing, local refinement	Mesh quality impacts accuracy and convergence
Time Stepping & Stability	Implicit, Explicit, Courant Number control	Accurate transient simulations, stability assurance	Stability and accuracy balance	Choosing proper time step size and convergence criteria

5. REFERENCES

[1] B. G. Thomas, "The 2002 Brimacombe Memorial Lecture Iron and Steel Society Modeling of the Continuous Casting of Steel-Past, Present, and Future."

[2] S. Louhenkilpi *et al.*, "Online Modelling of Heat Transfer, Solidification and Microstructure in Continuous Casting of Steel," in *IOP Conference Series: Materials Science and Engineering*, Institute of Physics Publishing, Jun. 2019. doi: 10.1088/1757-899X/529/1/012051.

[3] D. Mazumdar and R. I. L. Guthrie1, "The Physical and Mathematical Tundish Systems Modelling of Continuous Oasting," 1999.

[4] L. Zhang and B. G. Thomas, "State of the Art in Evaluation and Control of Steel Cleanliness," *ISIJ International*, vol. 43, pp. 271–291, 2003.

[5] K. Timmel, C. Kratzsch, A. Asad, D. Schurmann, R. Schwarze, and S. Eckert, "Experimental and Numerical Modeling of Fluid Flow Processes in Continuous Casting: Results from the LIMMCAST-Project," in *IOP Conference Series: Materials Science and Engineering*, Institute of Physics Publishing, Aug. 2017. doi: 10.1088/1757-899X/228/1/012019.

[6] K. Timmel *et al.*, "Experimental and numerical modelling of the fluid flow in the continuous casting of steel," 2013. doi: 10.1140/epjst/e2013-01804-5.

[7] A. Vakhrushev *et al.*, "Impact of Submerged Entry Nozzle (SEN) Immersion Depth on Meniscus Flow in Continuous Casting Mold under Electromagnetic Brake (EMBr)," *Metals (Basel)*, vol. 13, no. 3, Mar. 2023, doi: 10.3390/met13030444.

[8] Z. Liu, Z. Sun, and B. Li, "Modeling of Quasi-Four-Phase Flow in Continuous Casting Mold Using Hybrid Eulerian and Lagrangian Approach," *Metallurgical and Materials Transactions B: Process Metallurgy and Materials Processing Science*, vol. 48, no. 2, pp. 1248–1267, Apr. 2017, doi: 10.1007/s11663-016-0881-9.

[9] M. Iguchi, J. Yoshida, T. Shimizu, and Y. Mizuno, "Model Study on the Entrapment of Mold Powder into Molten Steel," *ISIJ International*, vol. 40, no. 7, pp. 685–691, 2000, doi: 10.2355/isijinternational.40.685.

[10] V. R. Voller and C. R. Swaminathan, "ERAL SOURCE-BASED METHOD FOR SOLIDIFICATION PHASE CHANGE," *Numerical Heat Transfer, Part B Fundamentals*, vol. 19, no. 2, pp. 175–189, Jan. 1991, doi: 10.1080/10407799108944962.

[11] A. Vakhrushev *et al.*, "Effects of the asymmetric and oscillating turbulent melt flow on the heat transfer and solidification inside the thin slab continuous casting (TSC) mold under the applied electromagnetic brake (EMBr)," in *Journal of Physics: Conference Series*, Institute of Physics, 2024. doi: 10.1088/1742-6596/2766/1/012196.

[12] H. Barati, M. Wu, S. Ilie, A. Kharicha, and A. Ludwig, "Numerical modeling of clog fragmentation during SEN clogging in steel continuous casting," *Powder Technol*, vol. 434, Feb. 2024, doi: 10.1016/j.powtec.2023.119307.

[13] P. Moin and K. Mahesh, "Direct numerical simulation: A Tool in Turbulence Research," *Annu Rev Fluid Mech*, vol. 30, no. Volume 30, 1998, pp. 539–578, Jan. 1998, doi: 10.1146/ANNUREV.FLUID.30.1.539/CITE/REFWORKS.

[14] B. E. Launder and D. B. Spalding, "The numerical computation of turbulent flows," *Comput Methods Appl Mech Eng*, vol. 3, no. 2, pp. 269–289, Mar. 1974, doi: 10.1016/0045-7825(74)90029-2.

[15] F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA Journal*, vol. 32, no. 8, pp. 1598–1605, 1994, doi: 10.2514/3.12149.

[16] "Turbulence models in CFD - RANS, DES, LES and DNS." Accessed: May 28, 2025. [Online]. Available: https://www.idealsimulations.com/resources/turbulence-models-in-cfd/

[17] H. Yang, S. P. Vanka, and B. G. Thomas, "Mathematical modeling of multiphase flow in steel continuous casting," 2019, *Iron and Steel Institute of Japan*. doi: 10.2355/isijinternational.ISIJINT-2018-743.

[18] "15.9. Pull Velocity for Continuous Casting." Accessed: May 19, 2025. [Online]. Available: https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/corp/v251/en/flu_th/flu_th_sec_melt_pull_velocity.html?q=porosity%20technique

[19] "6.1. Mesh Topologies." Accessed: May 30, 2025. [Online]. Available: https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/corp/v242/en/flu_ug/flu_ug_GridTypes.html

[20] A. Rabiee and A. Atf, "Control rod drop hydrodynamic analysis for a pressurized water reactor," *Progress in Nuclear Energy*, vol. 88, pp. 191–197, Apr. 2016, doi: 10.1016/j.pnucene.2015.12.007.