MOULD FLOW PREDICTION AND OPTIMIZATION[1]

L.J. Heaslip[2]
J.D. Dorricott[2]
J. Richaud[2]
J.P. Rogler[2]
W. A.Alves[3]

Abstract
Mould flow characterization of behavior has traditionally been performed by means of physical water modeling. In general, water modeling requires full-scale apparatus in order to provide reasonable similitude to the actual steel system, but physical modeling has certain limitations. These limitations include difficulties accounting for the pressure and thermal gradients that occur in the steel system. Recent advancements in computational fluid dynamics (CFD) improve the applicability of this tool in mould flow analysis. However, significant enhancements to commercial CFD software, such as customization of turbulence models and properly developed boundary layer analytic methods are required. The production of meaningful solutions requires a highly skilled and knowledgeable practitioner. Substantial experience with water models is necessary for model validation and training of the researcher. It remains essential that physical (water) modeling be used as a complementary tool to CFD analysis. CFD simulation in conjunction with water modeling has advanced steel flow control from tundish-to-mould and SEN design. These advancements include improved tundish-to-mould flow regulation, enhanced mould flow stability and symmetry, and reduced meniscus fluctuations and standing wave.
Key words: Mould flow; Water modeling; Continuous casting; Steel flow analysis

1 *XXXVII Steelmaking Seminar – International, May 21th to 24th, 2006, Porto Alegre, RS, Brazil*
2 *Advent Process Engineering, Inc. ,5035 North Service Rd., C-13, Burlington, ON, Canada, L7L-5V2.*
3 *Vesuvius Brazil, Rua Mexico No. 39, 06415-160 Barueri, Sao Paulo, Brazil*

INTRODUCTION

Since its inception in 1989, Advent Process Engineering has been deeply involved in the analysis of fluid flow phenomena as applied to the processing of liquid steel. Comprehensive physical (i.e. water) modeling has been conducted of fluid flow in virtually all aspects of steelmaking from furnace to the casting mould. In recent years, the continuous casting process has seen major innovative developments that have altered the entire landscape of the world's steel industry. High-speed thin slab casting was introduced and rapidly commercialized. Medium thickness slab casting has developed toward technical and commercial dominance in plate and heavy skelp markets. Most recently, direct strip casting has shown at least technical feasibility. Throughout these advancements, Advent has been active in the design, development and application of the new liquid steel flow technologies that are required to complement and further these developments.

In continuous casting, molten (liquid) steel is poured from the ladle to the tundish. From the tundish, the liquid steel is caused to flow into one or more water-cooled moulds where it begins to solidify. It is of fundamental importance to the performance of this process, from the perspectives of both productivity and quality, to understand mould flow behaviour. Traditionally, water modeling has been the tool of convenience for the investigation of fluid flow phenomena in the continuous casting process as measurements of fluid dynamics in a molten steel bath are highly problematic. The technical justification for water as an analogue for liquid steel is based on the similar kinematic viscosities of liquid steel and water[1]. However certain limitations of water modeling prevent the approach to exact similitude with liquid steel actual continuous casting moulds. These limitations include difficulties in accounting for thermal gradients and the presence of a solidifying shell, as well as the pressure gradients that occur in the steel system.

Rapid advancements in computing hardware and software have made viable the application of commercially available CFD (Computational Fluid Dynamics) packages to liquid steel flow analysis. Theoretically, such packages provide the opportunity to model the fluid flow in continuous casting with a greater degree of similitude. However, the determination of practical and valuable results generally requires significant enhancements to commercial CFD software, such as customization of turbulence models, properly developed boundary layer analytic methods and application-specific multiphase or VOF (Volume of Fluid) methods. In order to effectively and properly utilize CFD technology, so as to produce meaningful solutions, a skilled and knowledgeable researcher is absolutely essential. This researcher must also have the prior experience and understanding of mould flow behaviour that at this time can only be obtained through deep familiarity with water models. Thus, it is crucial that CFD modeling physical (water) modeling be used in a complementary and conjunctive manner. CFD analysis must first be capable of reproducing water model conditions and results related to the specific case study under investigation, before introducing any of the numerous complications and variables related to the analysis of the liquid steel system.

MOULD FLOW ISSUES

Water modeling provides a very convenient method to determine the influence of many parameters on the general mould flow behaviour as characterized by flow pattern visualization and suitable measurements such as sub-meniscus velocities and mould level fluctuations. Typical parameters of such a study may include:

- submerged-entry nozzle (SEN) geometry,
- SEN port submergence,
- flow control apparatus (stopper or slide gate),
- influence of deposition inside the casting channel referred as clogging,
- the typical casting parameters such as; casting speed and mould width,
- Argon injection into stopper or SEN.

The purposes of such studies vary from an analysis of process capability in terms of casting speed, casting section size or throughput, as well as the determination of the origin of specific cast product defects. Some examples[2] of issues related to mould flow are:

- narrow face steel shell thinning due to strong nozzle jet impingement and leading to possible breakouts,
- mould level instability, meniscus fluctuations leading to oscillation marks, surface cracks and mould powder entrapment,
- excessive sub-meniscus velocity that can tear off mould molten powder,
- vortexing on the meniscus that can lead to mould powder ingestion into the cast product,
- accumulation, solidification and sticking of mould powder to the mould forming slag ropes,
- poor thermal distribution in the meniscus region causing local steel freezing and meniscus hooks,
- gas bubble and particle entrapment in the steel shell in the lower regions of the mould.

MOULD FLOW UNDERSTANDING

Mould flow differs substantially in nature from the flow inside the pouring nozzle (SEN) connecting the tundish to the mould. However, it is the flow exiting the nozzle that drives the mould flow. Typically the nozzle bore velocity is 5 to 20 times greater than the sub-meniscus velocity. Flow exits the nozzle and enters into the mould, forming a penetrating jet that dissipates inside the liquid pool contained inside the solidifying steel shell. Momentum dissipation in the mould is governed by viscous effects involving the penetrating jet and eddies that are generated. These eddies are turbulent and wide ranging in scale. The overall mould flow pattern is generally re-circulatory in nature and in a slab mould can be represented as a single loop or a double loop on each side of the SEN depending upon many factors such as impingement angle, jet momentum and port submergence. The turbulent jet-driven re-circulating nature of mould flow behaviour makes this one of the most difficult types of flow to analyze and predict with confidence by CFD.

Water model observations of flow passing around an immersed cylinder of 50mm in diameter in the meniscus region reveal that the flow is not chaotic or fully turbulent. The local Reynolds number (R=Meniscus **V**elocity x immersed rod **D**iameter / kinematic viscosity of water) of the moving fluid around the immersed tube is typically lower than 350000.

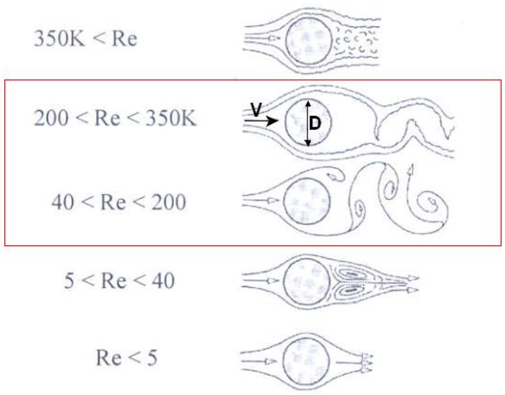

Fig. 1: Various Flow Regimes Governed by Re # for Flow Around a Cylinder

Mould flows are highly complex to model. Most of the elements of turbulent flows are encountered; large-scale unsteady structure, transitional flows and re-laminarization, rotational and swirling flows, separated and re-circulating flows, rapidly strained flows, cross flow and secondary flows, streamline vortices, free shear flows and thin and thick boundary layers. Such flows have extreme variation in local or cell Re (Reynolds Number). Figure 1 illustrates the influence of Re for the case of flow around a cylinder. It can be seen that various flow structures can be formed depending on Re and that for a wide range of Re the flow can include complex vortex or eddy shedding.

The first challenge of a CFD simulation is to capture accurately the average values of various parameters measured in the water model such as velocity, pressure, and meniscus shape. Then the second challenge is to reproduce the unsteady aperiodic motion of the water model flow. Transient models of flow using for example an LES (Large Eddy Simulation) or even a combined RANS (Reynolds-averaged Navier Stokes) with DES (Detached Eddy Simulation) model can be used. Since the mould flow also contains a very wide range of eddy scales, it is imperative to determine what turbulence scale can be practically computed versus what needs to be modeled. Water model results can provide some guidance in this regard.

TURBULENCE MODELING

A complete time-dependant solution of the exact Navier-Stokes (N-S) equations for turbulent flows in systems such as the continuous casting mould is not attainable at the current time, nor is this likely for some time to come. Two methods are available to transform the N-S equations so that small-scale turbulent fluctuating eddies do not have to be directly modeled; Reynolds averaging and filtering. The Reynolds-averaged approach is most common for general engineering calculations. This approach reduces the exact N-S equations to a simplified form known as the RANS (Reynolds-averaged N-S equations) and requires an eddy viscosity model (EVM) to account for the influence of turbulence on the effective viscosity. The EVM most often used is a two-equation model (k-ε, or k-ω for example). In the k-ε model, the following equations describe the conservation of k (1a) and ε (1b) as well as their relation to the turbulent viscosity (1c).

$$\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}=\frac{\mu_t}{\rho}S^2-\varepsilon+\frac{\partial}{\partial x_j}\left[\frac{1}{\rho}\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] \qquad (1a)$$

$$\frac{\partial \varepsilon}{\partial t}+U_j\frac{\partial \varepsilon}{\partial x_j}=\frac{\varepsilon}{k}\left(C_{1\varepsilon}\frac{\mu_t}{\rho}S^2-C_{2\varepsilon}\varepsilon\right)+\frac{\partial}{\partial x_j}\left[\frac{1}{\rho}\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] \qquad (1b)$$

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \qquad (1c)$$

Even with this relatively simple approach, the first difficulty is to correctly model the near wall region, which is important since solid walls are a main source of vorticity and turbulence (large variations of turbulence dissipation and local extremes of turbulent kinetic energy). Near the walls, such as the narrow face walls in a slab mould, flow separates and reattaches. This complex behaviour depends upon the prediction of the development of the turbulence near the walls. The k-ε EVM can be applied in the turbulent core region of the flow and through the log-law layer approaching the wall, but near wall boundary flows are known to be anisotropic and cannot be determined from the isotropic behaviour predicted by classical k-ε models.

In order to better reproduce near wall flow behaviour, a damping function approach can be introduced in the low Reynolds number regions. Algebraic functions (2c) are introduced to damp certain terms of the turbulent dissipation rate equation (2a) and eventually correct the behaviour of the eddy viscosity (2b). In most of early numerical simulations performed, the damping function used was the Launder and Sharma model. But many other models are available. The damping functions approach is illustrated by the following equations:

$$\frac{\partial \varepsilon}{\partial t}+U_j\frac{\partial \varepsilon}{\partial x_j}=\frac{\varepsilon}{k}\left(f_1C_{1\varepsilon}\frac{\mu_t}{\rho}S^2-f_2C_{2\varepsilon}\varepsilon\right)+\frac{\partial}{\partial x_j}\left[\frac{1}{\rho}\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] \qquad (2a)$$

$$\mu_t=\rho C_\mu f_\mu\frac{k^2}{\varepsilon} \qquad (2b)$$

where:

$$f_1=1,\quad f_2=1-0.3e^{-\mathrm{Re}_T^2},\quad f_\mu=e^{\frac{-3.4}{(1+0.02\mathrm{Re}_T)^2}},\quad \mathrm{Re}_T=\rho k^2\big/\mu\varepsilon \qquad (2c)$$

But when comparing predicted results with water model results, it was found that the k-ε model was overly diffusive in the turbulent core of the computational domain of the mould, and was not always sufficiently accurate in regions close to no-slip walls even when relying on damping functions. The jet flow is almost considered as rectilinear, which is not adequate to the type of flow observed in slab moulds. In addition, only the gradients of the stresses act on the jet flow (especially anywhere else other than in impact regions) and gradients are usually larger in the direction orthogonal to the streamlines. The EVM underestimates the prediction of secondary motions observed around the jet exiting the nozzle, which are given by the difference between normal stresses. In the nozzle port region, flow is not correctly captured. Considering that the viscous stresses, not their gradients enter into the turbulence production, and due to the large curvature of the streamlines exiting the port, the stresses in this region are very high. Consequently the model over-predicts turbulence production. Even with the application of Kato-Launder[3] modification, which reduces the tendency of k-ε models to over predict turbulence production in regions of large strain, i.e. regions with strong acceleration or deceleration such as the ports, the k-ε EVM is unable to accurately reproduce the overall average flow pattern in the water model.

Other two-equation EVM have been examined; including a variation of the k-ε model called the RNG model and the standard k-ω[4] model. The most successful of these models was a variation of the k-ω model called the SST-k-ω model[5]. This model is attractive since no special near wall treatment is required. The model is designed for application in both turbulent core and wall regions and is sufficiently robust for use with coarse meshes. In order to better correlate the CFD simulation to the water model, some of the coefficients determining the limit between the low Re region and high turbulence regions were modified. This tuning helped to reduce the need for extremely refined grid along the walls of the mould. This is a great advantage considering that CFD simulation of the entire casting channel from tundish-to-mould already requires very large meshes. The SST-k-ω has been adapted by means of user-defined functions for turbulent viscosity to further improve water model correlations.

Figure 2 shows a comparison of water modeling and corresponding CFD analysis. The nozzle design being studied had relatively large ports and the CFD analysis with the k-ε EVM with low Reynolds number correction does not correctly capture the swirling flow inside and exiting the port. The prediction of this CFD analysis did not reveal the presence in the jet of the large eddies that are evident in the water model snapshot. Furthermore, the predicted jet is overly diffusive and does not penetrate into the mould pool in the manner seen in the water. When simulating wide slab, the predicted jet flow does not even reach the narrow faces before jet curling occurs and this is not in accordance with the water model. Using the modified SST-k-ω EVM, the jet swirling found inside the SEN port in the water model has been reproduced and eddies propagating out of the port in the jet are also properly captured. The combined approach of water modeling and CFD analysis has provided the opportunity for both development and validation of the modeling methods.

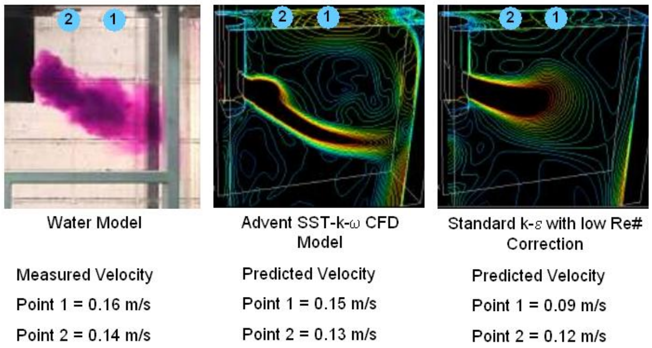

Fig. 2: Comparison of Mould Flow in Water Model to that predicted by Various CFD Models

EXAMPLES

Successful adaptation of CFD analysis methods so as to be in reasonable accordance with water models allows more confident application of CFD analysis to actual liquid steel flows. In the first instance, CFD application to the liquid steel flow may mean simply changing the properties of the liquid from those of water to those of liquid steel and modifying boundary conditions including thermal boundary conditions appropriately. Further development of a particular CFD analysis might include the effects of solidification or prediction of inclusion paths, which are difficult if not impossible to include in water modeling. Regardless of the further CFD analysis that is desired, the step of water model validation provides an increased surety of meaningful results.

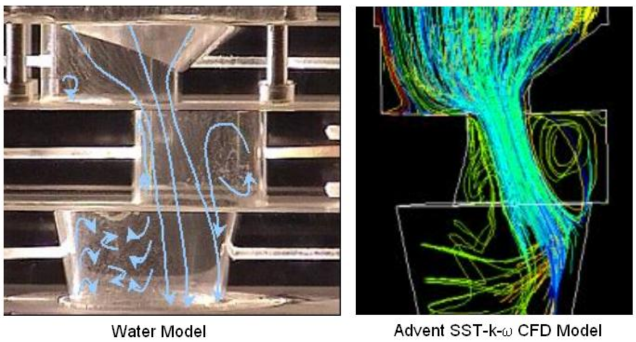

Fig. 3: Comparison of Flow in 3-Plate Gate Stack-up in Water Model and that predicted by Advent SST-k-ω CFD Model

Slide Gate

Figure 3 compares a CFD analysis of steel flow through an offset tundish slide-gate valve to a water model[6]. It has been found that conditions at the inlet and exit boundaries are critical for determining a correct flow inside the casting channel of the valve. The measured flow output passing through a certain tundish gate opening was compared to the predicted flow throughput given by the numerical solution. Then the model user defined functions were modified to reproduce the same result as measured in the water model. Once the numerical simulation produced satisfactory results, the physical properties of the fluid were changed to that of liquid steel and the simulation was recomputed.

If the inlet boundary condition is located in a region where the turbulence level is not strongly influencing the flow inside the channel, then the flow in critical areas such as the regulation is correctly calculated. Of particular interest in this study was pressure distribution in the gate when casting steel and the potential for air leakage and cavitations.

Narrow Face Mould Powder Accumulation (Roping)

A study conducted for a North American steelmaker highlights the influence of the mould flow near the narrow faces. Flow in the mould has a substantial impact on the flow at the meniscus and as a result effects mould powder melting along the meniscus. If adequate heat is not dispersed along the whole of the meniscus, mould powder may not melt and consequently may agglomerate in stagnant zones. Fig. 4 is an example of a mould powder agglomeration removed from the meniscus. This agglomeration is a result of a stagnation region that wraps around the corners of the mould where the narrow face meets the broad face. This was shown to promote rope formation in this region, which can cause transverse cracks and depressions on the slab corners and narrow faces.

Fig. 4: Example of Rope Removed from the Mould

Liquid steel flow was computed with the Advent-modified SST-k-ω EVM to identify the mechanism by which this rope forms. As shown in Fig. 5, the strong upward flow climbing the narrow face detaches from the face and curls inwardly and impinges strongly upon the meniscus some distance in from the narrow face. Accordingly the general mould flow bypasses the corners of the meniscus and consequently a re-circulating zone is formed. The strong flow that impinges on the meniscus elevates the surface locally and as a result pushes mould powder toward the narrow face wall and builds up where the re-circulating dead zone occurs in the mould. Since the general flow of hot steel bypasses the mould corners and there is enhanced heat transfer in the corners, this region becomes relatively cold and thus there is insufficient heat to melt the mould powder in this zone, which results in the rope formation of agglomerated mould powder previously shown in Figure 4.

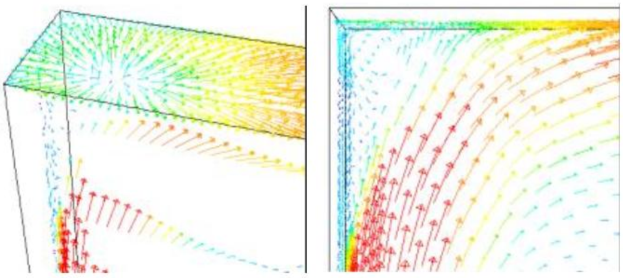

Fig. 5: Mould Flow Resulting in a Re-circulating Dead Zone on the Meniscus

Off-Corner Erosion

In thin slab casting, the erosion on the broad faces of the nozzle can be observed. Both water model and numerical simulation show the influence of vortexing in the region where the liquid flows through a narrow passage between the nozzle and the mould.

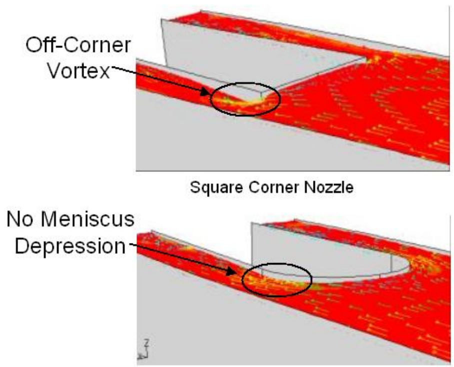

Fig. 6: Effect of Exterior Nozzle Design on Surrounding Meniscus Depression & Off-Corner Vortexing

Q.He proposed a vortex formation mechanism[7] in which vortex occurrence results from bias flow along the meniscus, i.e. the metal flow from the narrow face towards the nozzle on one side is stronger than the opposite side, thus generating a vortex in the weak region. Although bias in the flow along the meniscus definitely exacerbates vortexing, off-corner vortexing in a thin slab bulge mould has been observed and predicted to occur in the absence of bias flow. Vortex formation in the off-corner region of the nozzle was examined using the Advent-modified SST-k-ω EVM. In the absence of bias flow, vortex formation was found to depend on nozzle shape, the gap between the nozzle and the mould and the sub-meniscus velocity. When sub-meniscus velocity exceeds 0.1m/s, a bow wave forms on the leading edge of the SEN and induces an off-corner depression along the broad face of the SEN. Fig. 6 illustrates the effect of the cross-sectional exterior geometry of nozzle, as the square cornered nozzle induces a much larger depression as compared to the rounded corner nozzle. In the region of the depression, flow becomes detached from the nozzle; vortices are shed along this detached region and subsequently coalesce downstream to form a larger vortex along the refractory wall. The vortex observed in the off-corner location can entrain mould powder along the nozzle but mostly introduce turbulence in the chemical boundary layers at the refractory/steel/mould-powder interface. Both mechanical and chemical erosion takes place and local erosion can lead to a hole formation as shown in Fig. 7. Consequently mould powder can be entrained into the nozzle flow and transported deep into the mould resulting in product defects.

Fig. 7:Examples of Erosion on the Exterior of Nozzles From Off-Corner Vortexing

CONCLUSIONS

CFD simulation is an evolving tool that may provide an enhanced understanding of phenomena unachievable by means of physical (water) modeling alone. The necessity of physical modeling to verify CFD solutions and adjust turbulence models accordingly has been confirmed. CFD simulation and correlation with the water model allows for validation of the turbulence models before they are applied to the final steelmaking solution. Examples have been shown in which CFD simulation of liquid steel flow provided meaningful solutions that have been used to further the understanding of fluid flow and allow subsequent advancements in the technology of continuous casting.

REFRERENCES

1 L.J. Heaslip, A. McLean, and I.D. Sommerville, Continuous Casting, ISS-AIME, 1983, Vol. 1, pp 67-71.
2 B. Thomas, ICS Proceedings 2005, p847-861.
3 Kato, M. and Launder, B.E., Proc. 9th Symposium on Turbulent Shear Flows, 1993, pp. 10.4.1-10.4.6.
4 D.C. Wilcox, AIAA Journal, Vol. 26, 1988, pp. 1414-1421.
5 F.R. Menter, AIAA Journal, Vol. 32, 1994, pp. 269-289.
6 P.D. King et al., ISS Tech Proceedings 2003, pp 265-282.
7 Q. He, ISIJ International, Vol. 33, No.2, 1993, pp. 343-345.