Thomas, B.G. and P. Vanka; "Study of Transient Flow Structures in the Continuous Casting of Steel", 2000 NSF Design & Manufacturing Grantees Conference, Vancouver, Canada, Jan. 5-8 2000, 14p.

Study of Transient Flow Structures in the Continuous Casting of Steel

B.G. Thomas and S.P. Vanka

University of Illinois at Urbana-Champaign
Mechanical and Industrial Engineering.
1206 West Green Street, Urbana, IL 61801
Ph: 217-333-6919, 217-244-8388; Fax: 217-244-6534;
Email: bgthomas@uiuc.edu, s-vanka@uiuc.edu

Abstract: In continuous casting of steel, plant observations have found that many serious quality problems are directly associated with the flow pattern in the mold. Previous studies have generated understanding mainly through numerical simulations using time-averaged turbulence models. However, many problems are intermittent and the essential transient nature of the flow may be important to their formation. To obtain further understanding of these important transient turbulence processes, this project aims to directly compute the evolution and dynamics of the large scale turbulence structures. Accurate numerical schemes and parallel computers are being applied to solve the governing fluid flow equations using a Large-Eddy Simulation (LES) approach. Computations are also performed using traditional K-ε models in order to evaluate their accuracy and to examine more cases. The ultimate goal is to generate deeper understanding of how costly defects form and to find improvements in design and operating conditions in continuous casting that can avoid them.

Introduction: Continuous casting is the predominant way by which steel is produced in the world. Continued viability of the high-volume-low-profit-margin steel industry depends upon improved efficiency and consistent quality of the steel production.[1] Plant observations have found that many serious quality problems are directly associated with the flow pattern in the mold [1]. Defects caused by non-optimal fluid flow are even more important to the nearer-net-shape thin-slab casting processes, which are starting to transform the industry [2]. Some understanding of this flow region can be obtained through numerical simulations which use time-averaged turbulence models. The next step to obtain more reliable predictions of this transient turbulence process is to directly compute the evolution and dynamics of the large-scale turbulence structures. The current research is concerned with such computations, using accurate numerical schemes and parallel computers to solve the governing fluid flow equations.

A schematic of part of the continuous casting process is depicted in Figure 1. Steel flows through the "tundish," and then it exits down through a ceramic Submerged Entry Nozzle (SEN) and into the mold. Here, the steel freezes against the water-cooled copper walls to form a solid shell, which is continuously withdrawn from the

---

[1] Today, US produces around 80 million tons of steel per year. The net cost per ton of scrapping is about $100 per ton. Even if a fraction of one percent of scrap is avoided due to improving the process, the savings is still significant.

Schematic of continuous casting tundish, SEN, and mold

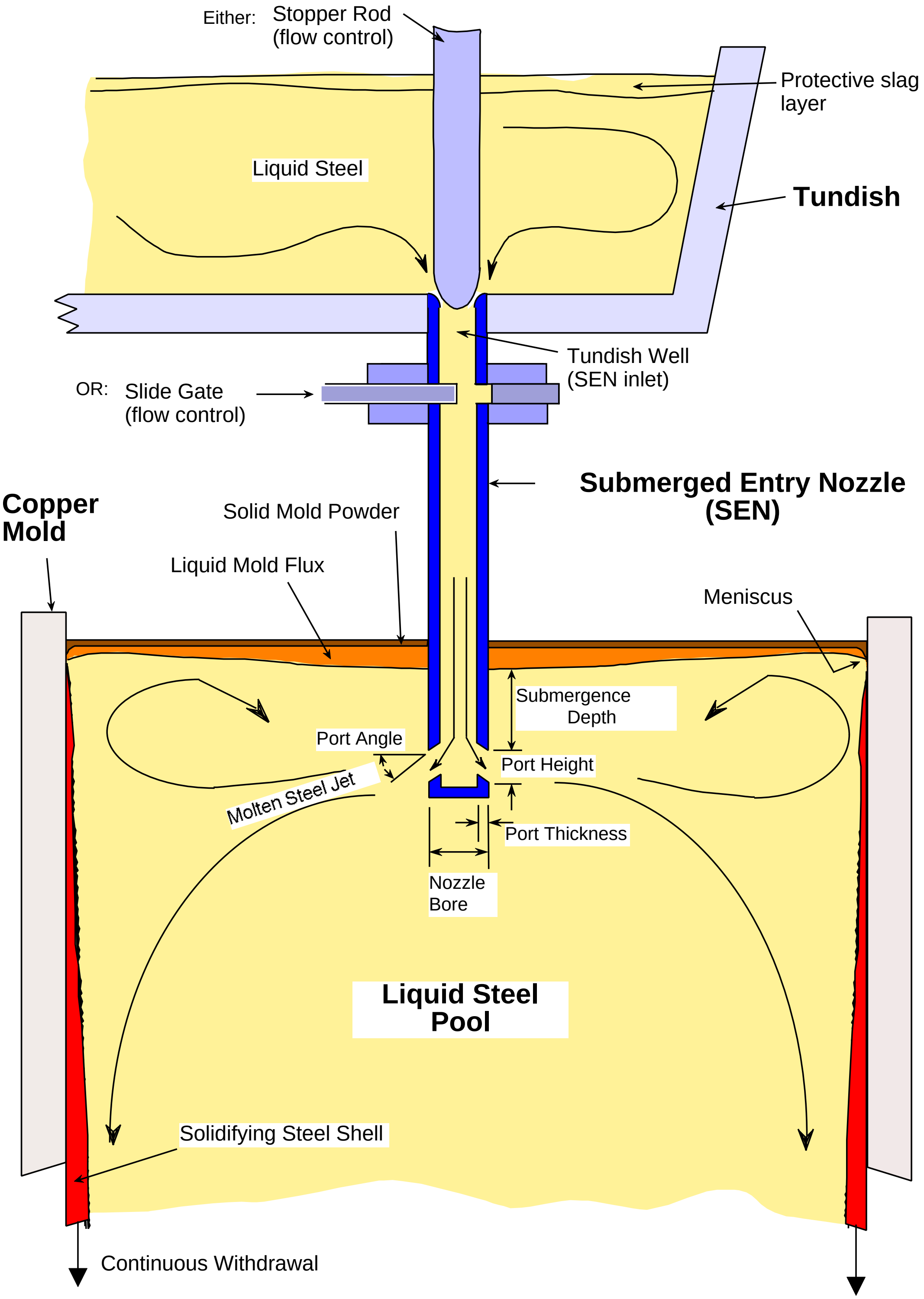


Figure 1. Schematic of tundish and mold region of continuous casting process

bottom of the mold at a "casting speed" that matches the flow of the incoming metal. The primary reason for submerging the nozzle is to protect the molten steel from re-oxidation as the steel is delivered from the tundish to the mold. Argon gas is injected into the nozzle to help prevent clogging with alumina inclusion deposits. The submerged nozzle also has an important influence on steel quality through its effect on the flow pattern in the mold. The nozzle should deliver steel uniformly into the mold while preventing problems such as surface waves, meniscus freezing, and crack formation.

Unsteady flow features play an important role in the continuous casting of steel, yet have received relatively little attention. In recent years, with the development of fast computers, it has become possible to significantly improve turbulent flow predictions by resolving the large scales of transient and turbulent flows [3-4]. These simulations, known as Large-Eddy Simulations (LES) lie in-between the approaches of Direct Numerical Simulations (DNS) and the Reynolds-averaged approach. In LES, the dominant, energy containing scales of motion are accurately resolved and the small scales are modeled. The premise of LES is that the small scales of turbulent motion are nearly isotropic and universal across different flows. Therefore, the effects of the small scales can be modeled relatively more accurately compared to modeling all the scales by a single model. In recent years, LES has been successfully applied to several flows.

In the present paper, we discuss some recent results of flow and heat transfer for four different parts of this project:

1) Two-phase flow in tundish nozzles
2) Unsteady flow in the mold region
3) Fluid flow and heat transfer in mold
4) Heat transfer in the impingement region

Technical Approach: Two different computational models of fluid flow are used in this work.

Firstly, the Reynolds-averaged approach was used to simulate the three-dimensional time-averaged two-phase flow and heat transfer fields in both the nozzle and mold regions. These models were developed using the K-ε turbulence model in the commercial package CFX.

Secondly, Large-Eddy Simulation models of transient flow in the mold and impinging regions have been developed. This computer program, LES3D, integrates the three-dimensional unsteady incompressible Navier-Stokes equations using an explicit fractional step algorithm. Further, in order to take full advantage of parallel computers, the algorithm has been implemented with a general domain decomposition strategy. Each sub-domain of the flow can be calculated separately on an individual processor with data interfacing at the sub-domain boundaries. Advantage is taken of the Message Passing Interface (MPI) standards to ensure portability across a variety of parallel computers, including shared and distributed memory machines.

In conjunction with the proposed modeling work, experiments are performed to measure the flow fields in water models, as well as in an operating steel caster. In addition to providing additional insight into the flow phenomena, these experiments are even more important to validate the mathematical models, so that subsequent parametric studies can be calculated with confidence. Measurements on both physical water models and in operating continuous casting in the plant have been obtained at several of the co-sponsoring steel companies, including Armco, Inc., and LTV Steel.

Results: The current results from this study are presented in the following four sections.

Two-Phase Flow in Tundish Nozzles:

The tundish nozzle has an important influence on steel quality through its effect on the flow pattern in the mold. Argon is commonly injected into the tundish nozzle to avoid nozzle clogging. It also affects casting operation and product quality by changing the flow pattern in the nozzle and mold. In this part of the project, a three-dimensional finite difference model is being applied to model the multi-phase, steady-state turbulent flow in continuous casting tundish nozzles under a wide range of geometries and conditions.

To validate the model, flow fields near the nozzle port outlets were measured with PIV (Particle Image Velocimetry) in a 0.4-scale water model at LTV Steel at Cleveland Ohio. These velocities were compared with predictions from the steady-state turbulent flow model. Reasonable agreement was achieved, as shown in the example given in Figure 2. Extensive parametric studies using the validated model have been performed to investigate the effects of many different variables on the flow pattern. These variables include the process variables (such as casting speed, tundish liquid level, slide-gate opening and orientation), argon injection parameters (such as gas injection flow rate, bubble size, gas injection area and locations), and the geometric parameters of the nozzle (such as the port angle, port height, port width, bottom shape).

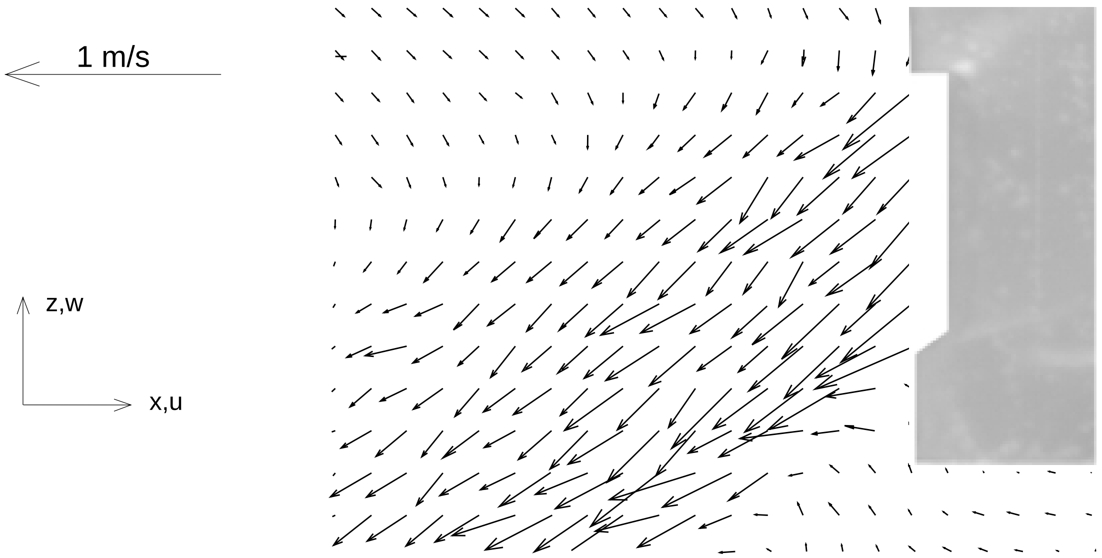
The numerical modeling results were converted into trends that correspond with real-life operating conditions, where several variables change simultaneously. To achieve this, an advanced multivariable curve-fitting model has been developed. For example, Figure 3 shows how the lowest pressure in the nozzle changes with the casting speed assuming variable gate opening under the standard operating conditions of fixed tundish liquid level and fixed argon injection rate. It can be seen that detrimental negative pressure is obtained at low casting speeds (below about 1 m/min for this nozzle, depending on tundish depth). Thus, smaller nozzle bores should be used for low casting speeds. During ladle exchanges, casting speed should only be slowed down if the tundish level is also lowered. Results such as these should help design standard operating conditions that minimize defects, such as the inclusions created when air aspiration occurs while operating at negative nozzle pressure.

Unsteady flow in the mold region:

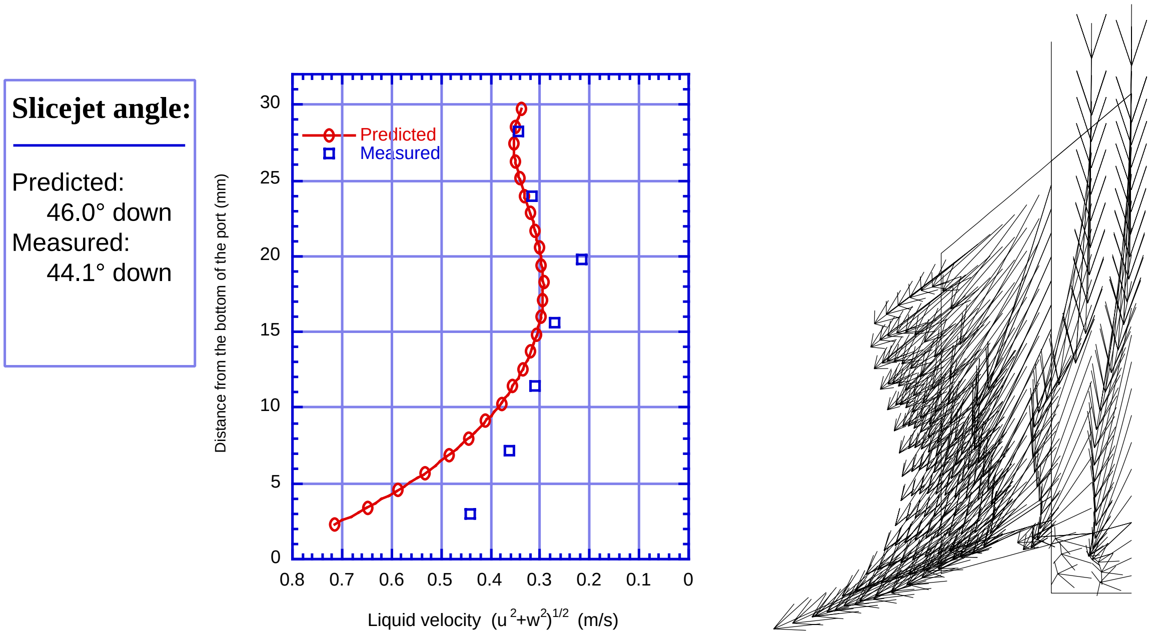
Transient flow phenomena in the mold may be very important to the generation of quality problems, such as surface level fluctuations, inclusion and bubble entrainment. To study these phenomena, a large-eddy simulation model is being developed. This model is being validated using PIV (Particle Image Velocimetry) measurements, done on a 0.4-scale water model of the mold (at LTV Steel, Cleveland, Ohio.

The LES model currently involves the following simplifications. Firstly, the inlet to the mold is approximated as fully-developed turbulent flow from a square duct, assuming that the actual profile at the inlet of the water model dissipates a short distance from the inlet, thereby not affecting the flow features in the mold significantly. Secondly, the top surface is approximated as a free slip boundary instead of the free surface that is present in the water model. This is justified as level fluctuations are found to be insignificant and the surface profile is almost flat.

Examples of the comparative study are given in Figure 4 (instantaneous) and Figure 5 (time averaged). The major findings of this comparative study are as follows.



(a) PIV measurement of flow field near the nozzle port (center plane)



(b) Velocity profile and average slice jet angle at the nozzle port

(c) Two-phase model prediction(CFX)

Figure 2 Comparison of prediction from the two-phase modle(CFX) with PIV measurements at the center plane of the nozzle port (Case A5: Q_{liq} = 14.2 gal/min, Q_{gas} = 1 scfh)

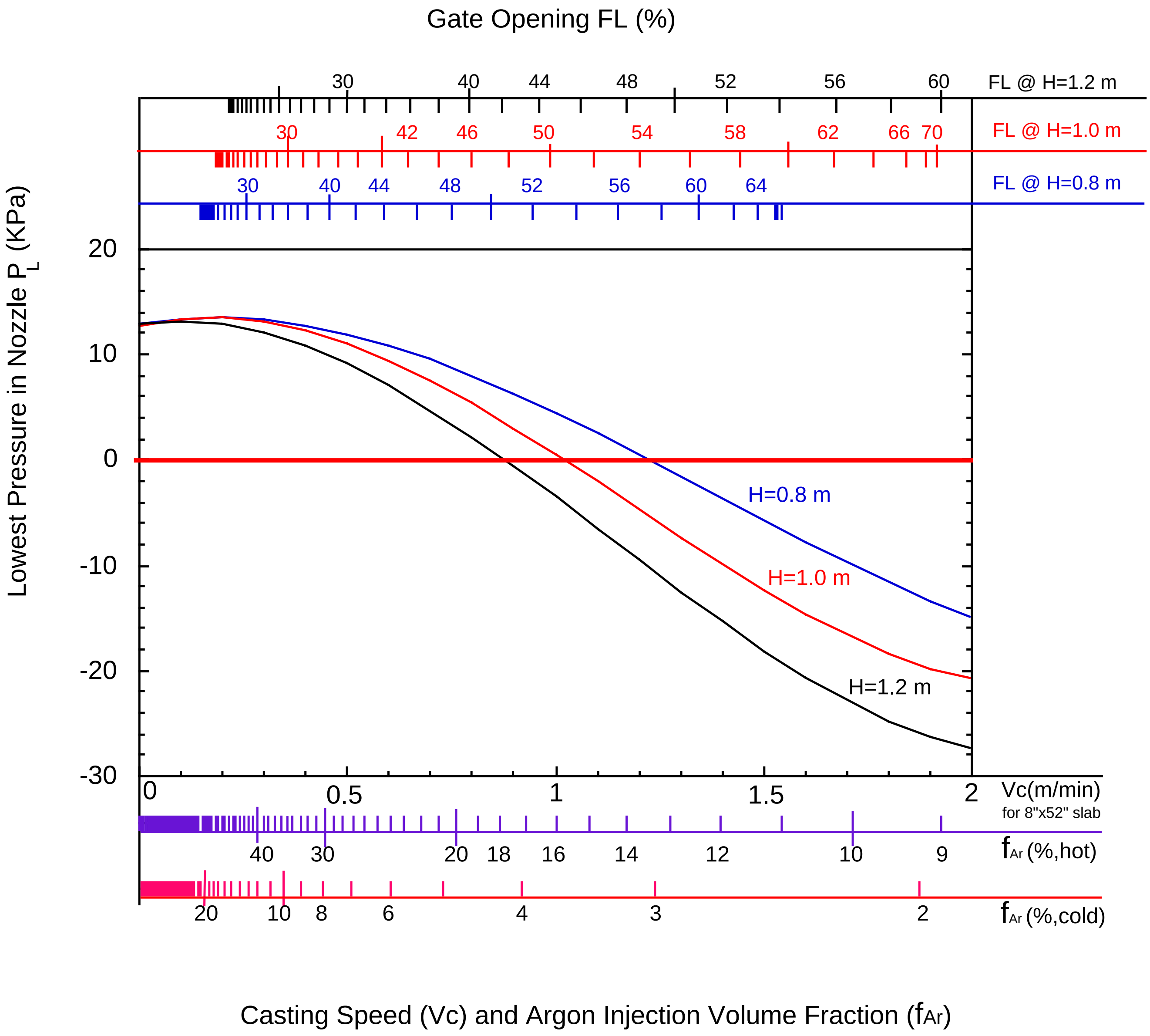


Figure 3 Lowest pressure in nozzle vs. casting speed at constant bath depth (HPQF model prediction)

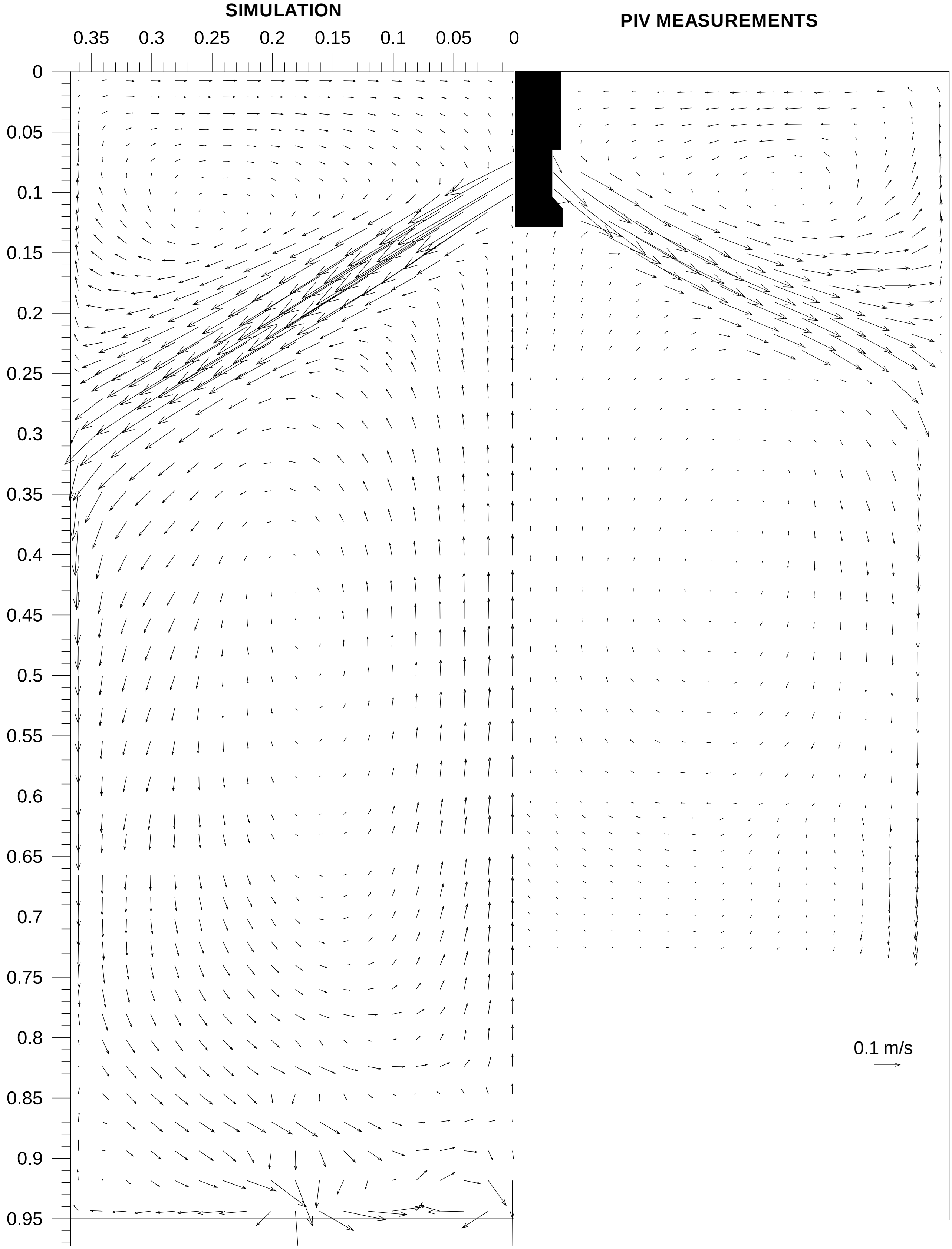


Figure 4 Comparison of time averaged simulation and PIV vector plots

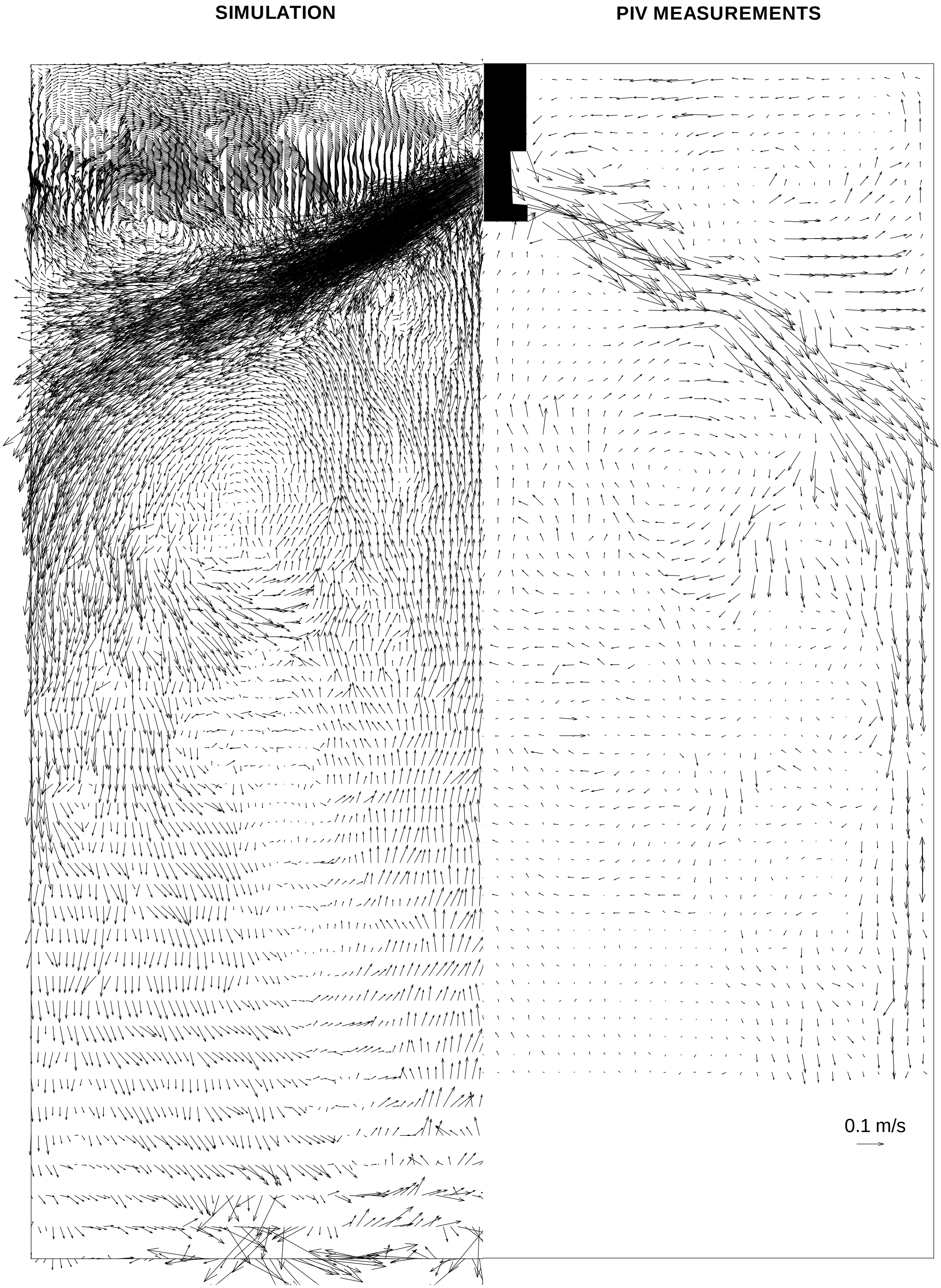


Figure 5 Comparison of time averaged simulation and PIV vector plots

The inlet condition is of considerable significance to the flow in the mold. The swirl at the port outlet persists until halfway across the mold. The experimental jet has considerable in and out of plane motion as compared to the simulation which has an inclined fully-developed turbulent square duct flow as the inlet condition. These two together cause the experimental jet to bend so that it impinges nearly horizontally on the narrow face.

The flow near the top surface in the experiment varies by more than 100% of its mean value. The measurements reveal high frequency variations (~1.5 Hz) which are also seen in the simulation. A typical signal also contains a low frequency component (time period of the order of 45s). This component is not seen in the simulation and is speculated to be due to the wide variations in the depth of penetration of the experimental jet which is also not seen in the simulation. This feature may be of considerable significance to shear entrainment of the liquid flux.

Although the geometry and inlet from the nozzle port are symmetric, there is considerable, persistent, asymmetry between the two lower rolls in the experiments. The flow in the lower rolls is not stationary but consists of a sequence of flow phenomena which repeats chaotically. One of the flow features involving a short-circuit of the downward moving flow with the upward moving one is seen in both experiment and simulation, suggesting that it is not dependent on input condition but might be caused by pressure instabilities or other small disturbances in the flow field. This feature may be important for particle and bubble entrapment.

Fluid Flow and Heat transfer in Mold:

Three-dimensional models of fluid flow and heat transfer in the mold are being developed using the K-ε turbulence model in CFX, using input conditions from models of the nozzle. To verify the simulation results of heat transfer, temperatures in liquid pool were measured in an operating thin slab caster and were compared with CFX simulation of these cases. To verify the simulation results of fluid flow, PIV water modeling experiments were done and compared with both K-ε model and LES model results.

Figure 6 shows an example comparison of both models with measurements along the jet direction. Both models are qualitatively correct. However, the Reynolds average model appears to have slightly larger turbulence dissipation, leading to lower velocities. The LES simulation, on the other hand, slightly overpredicts the velocity, probably due to the inaccurate inlet condition.

Gas is commonly added to prevent clogging of nozzle. In addition, the gas bubble size was found have a great influence on the flow pattern. The maximum gas penetration depth is found for the 1mm diameter bubbles case (Table 1). Either smaller bubbles or larger bubbles have better flow pattern. It is believed that, in practice, bubble size is normally over 1 mm. Thus, it is likely that the smaller bubbles are more often entrapped.

Table 1 Maximum gas penetration depth

Bubble Size	0.5 mm	1.0 mm	2.0 mm
35".min casting speed case	0.791 m	>3 m	0.221 m
55"/min casting speed case	--	>3 m	0.220 m

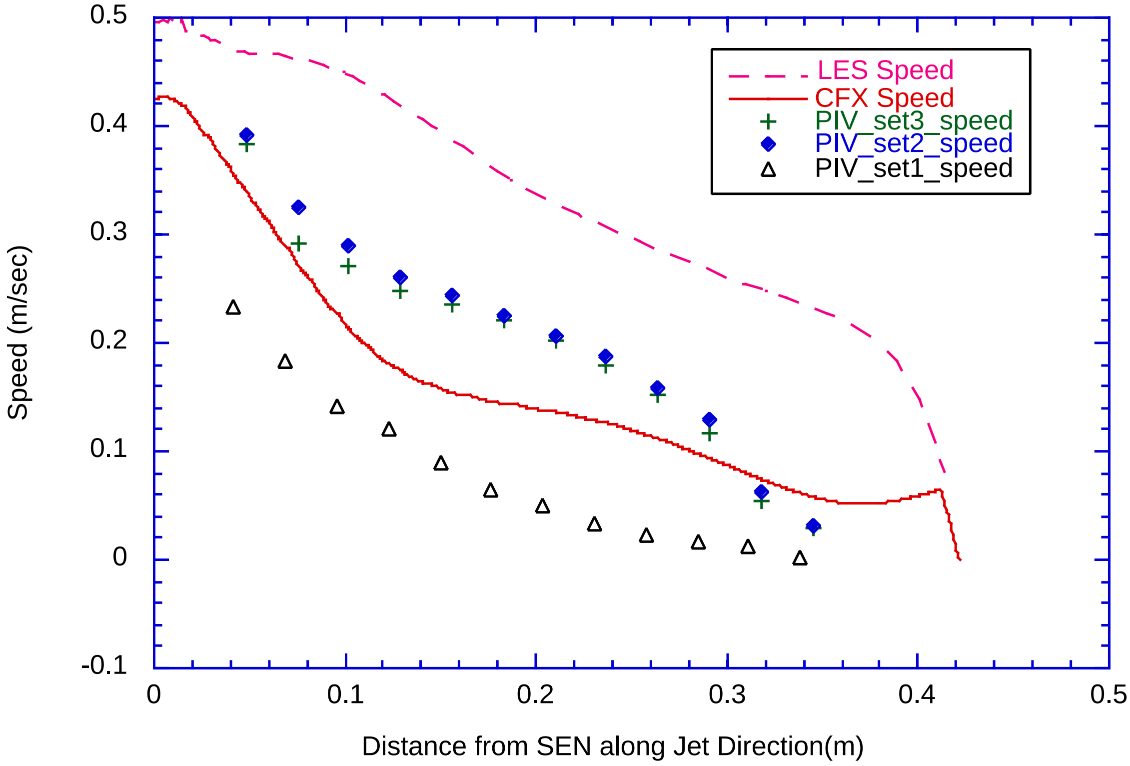


Figure 6 Comparison of CFX, LES simulation and PIV measured velocity

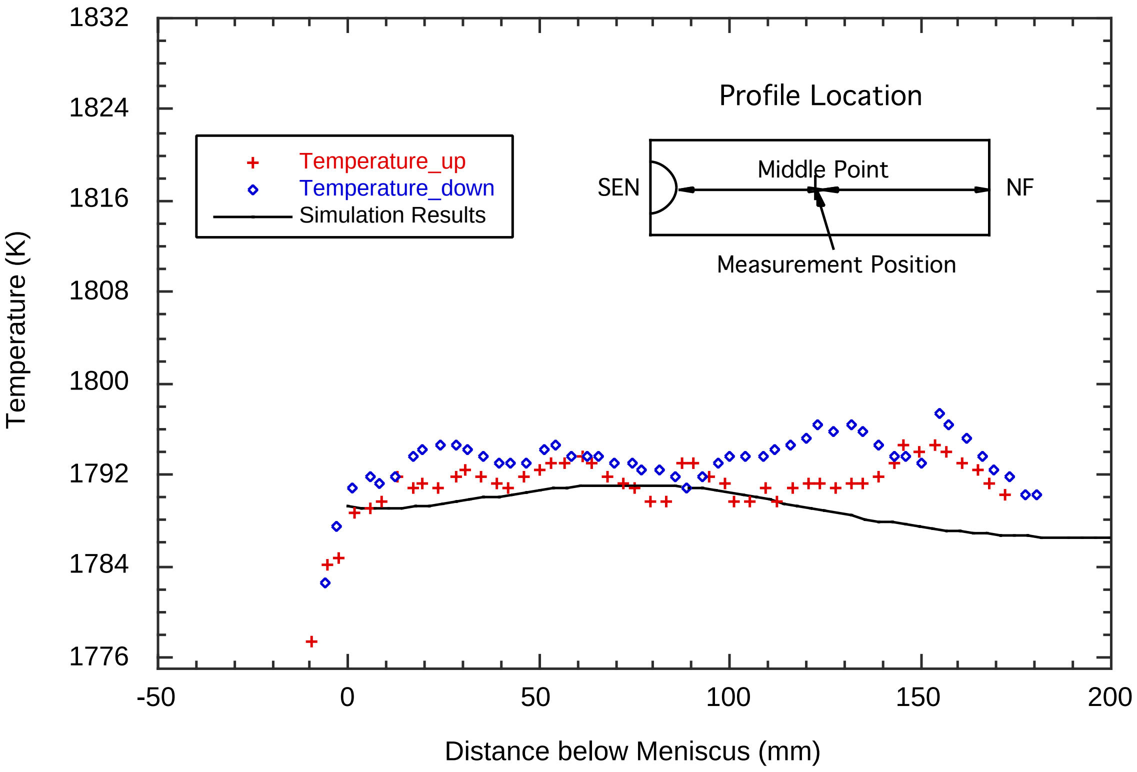


Figure 7 Comparison of Simulation and measured superheat in liquid pool

At high casting speed case, steel flow speed in the nozzle is faster, so bubble size should be smaller (based on previous work that found bubble size is inversely proportional to flow speed in the nozzle). This means there is higher probability of gas entrapment at high casting speed case, which matches findings in practice.

Temperature in the liquid pool affects heat flux to the solidified shell, which causes temperature differences, leading to thermal stresses and even cracks in the solidified shell. An accurate prediction of temperature in liquid pool is thus important for steel quality.

To verify accuracy of CFX prediction of temperature in liquid pool, temperature in practical caster was measured with a vertical thermocouple probe and compared with the CFX prediction. Fig. 7 shows that both the measurement and simulation become reasonably constant just below the meniscus. The dimensionless temperature θ, defined as ($\theta = \frac{T - T_{liquidus}}{T_{pouring} - T_{liquidus}}$) is about 0.26 in the simulation, which compares reasonably with 0.29 in the measurement. Comparisons in other positions also reveal similar agreement. This means that the CFX K-ε model is reliable in modeling temperature in liquid pool.

In the heat transfer simulation, the medium bubble size (1mm) case produces the highest temperature at the meniscus corner. For 2.0 mm bubbles, the high casting speed (55”min) case has higher temperature at the meniscus corner than the low casting speed (35”/min) case. Lower temperature at the meniscus corner increases the likelihood of subsurface hooks, which lead to sliver defects. This result is consistent with the observation that there are more sliver defects in the low casting speed (35”/min) case than at high casting speed (55”/min).

Heat transfer in the impingement region: Heat transfer in the flowing liquid in the continuous casting mold is important to predict the solidification rates of steel in the mold and to understand the formation of defects. Large-Eddy simulation models are being used to understand the fluid flow and heat transfer phenomena in impinging steel jets.

Heat transfer in flowing steel is difficult both to predict and to verify with measurements. This is because the flow is highly turbulent and molten steel has an intermediate Prandtl number (0.2) that has received little previous attention in the literature. In addition, the most important heat transfer is due to confined jet impingement against the walls, which involve non-parallel flow that is difficult for standard wall laws.

An LES simulation of a turbulent jet impinging normally on a flat plate has been studied and compared with experimental results of Holworth and Gero [5]. The computational domain is shown in Figure 8. A typical instantaneous flow field is shown in Figure 9 for the region near the point of jet impingement.

The mean Nusselt distribution is studied for several Reynolds numbers and compared with both the experimental data and an empirical correlation [6]. The mean Nusselt number distribution is provided in Figure 10. There is satisfactory agreement between the experimental data and the numerical results at Reynolds number of 5000. The difference in mean Nusselt number at the impingement point (r=0) is only 0.6%. The difference between the numerical and empirical values is only 0.8%. There is a slightly bigger difference at Reynolds number of 10000 (9% and 12%). The predicted distribution of the Nusselt number agrees with both the experimental and empirical curves.

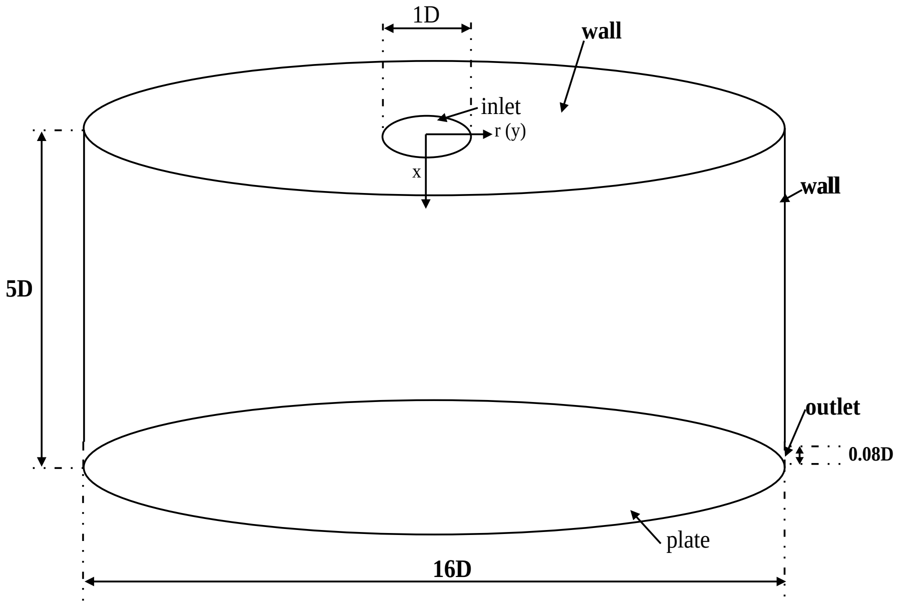


Figure 8 Domain of the numerical simulation

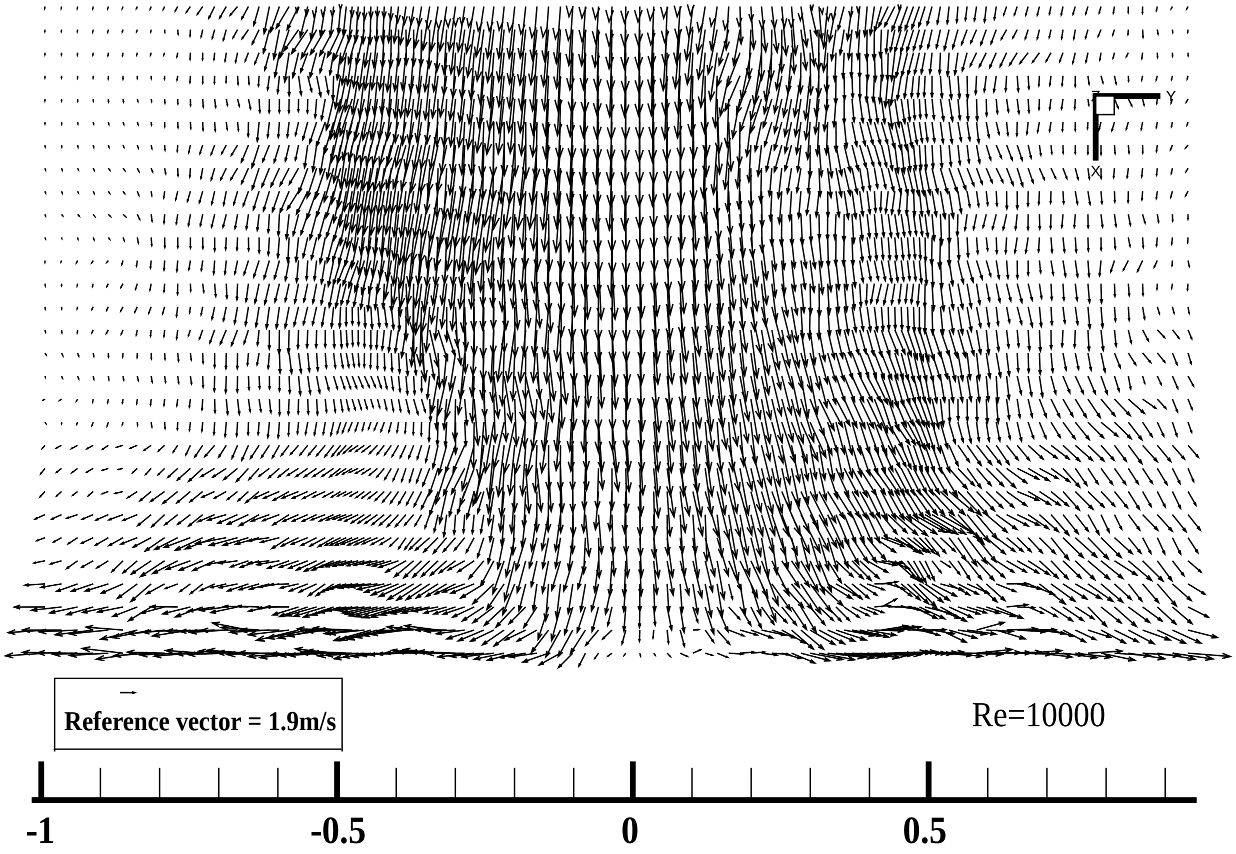


FIgure 9   Instantaneous velocity vector plot at the impingement region (t=12s)

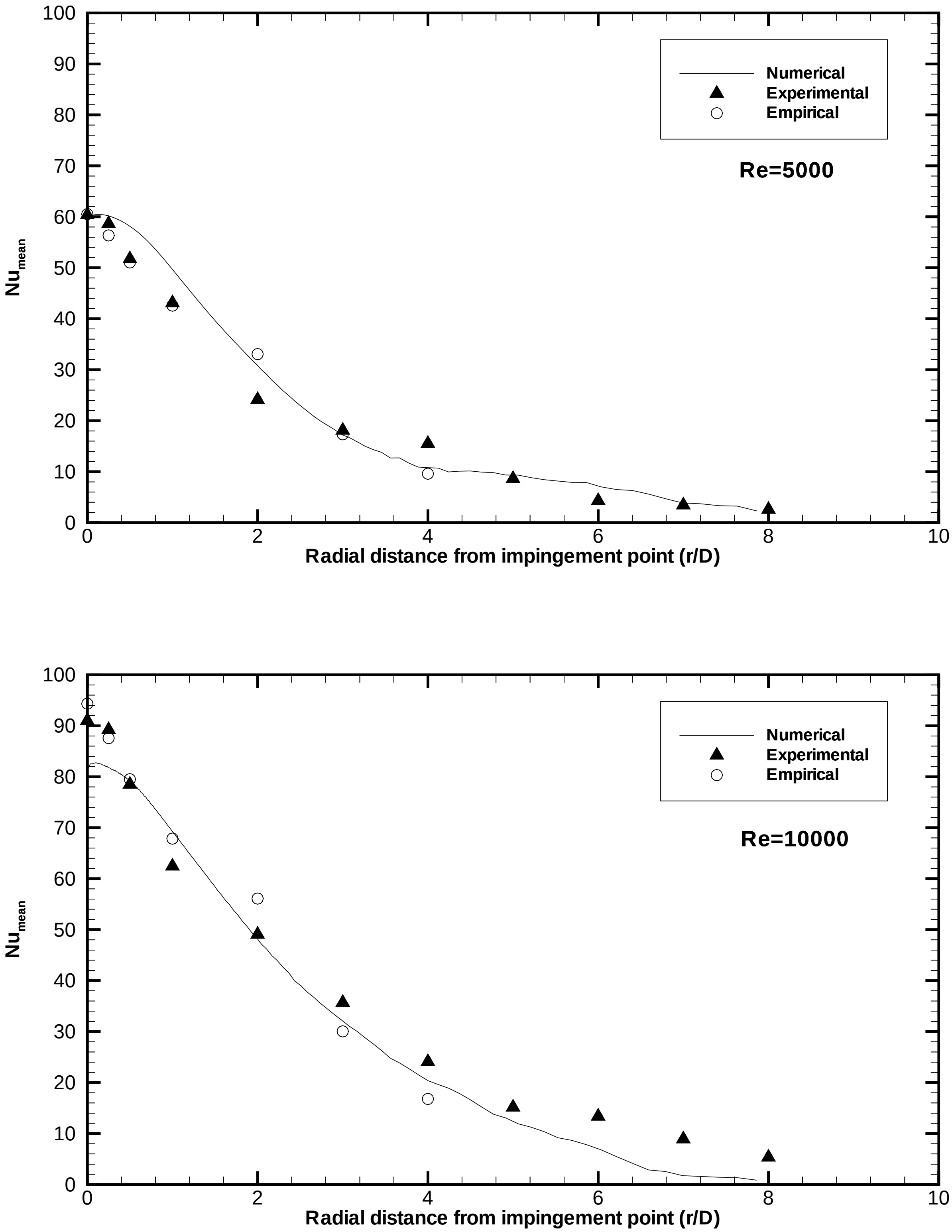


Figure 10 Radial distribution of the mean Nusselt number for various Reynolds numbers

After validating this model in a continuous casting domain and conducting grid dependency studies, reliable heat transfer predictions in the actual process will be obtained. The results obtained from studies with this model will provide valuable insights into defects in the process.

Further details on the work introduced here are given in Reference 7.

References:

1. J. Herbertson, O.L. He, P. J. Flint, and R. B. Mahapatra: in 74th Steelmaking Conference, Iron & Steel Society, Warrendale, PA., pp. 171-185, 1991.
2. T.A. Honeyands and J. Herbertson, "Oscillations in Thin Slab Caster Mold Flows", 127th ISIJ Meeting, March, 1994, Tokyo, Japan.
3. R. Rogallo and P. Moin, "Numerical Simulation of Turbulent Flows", Ann. Rev. Fluid Mechanics, vol. 16, p. 99, 1984.
3. U. Schumann, "Subgrid Scale Model for Finite Difference Simulations of Turbulent Flows in Plane Channels and Annulii", J. Comput. Phys., vol. 8, pp. 376-404, 1975.
4. Huang, X., and B.G. Thomas, "Modeling of Transient Flow Phenomena in Continuous Casting of Steel", Canadian Metallurgical Quarterly, 1998, vol. 37, no. 304, pp. 197-212, 1998.
5. Hollworth, B. R. and Gero, L. R. "Entrainment Effects On Impingement Heat Transfer, Part II – Local Heat Transfer Measurements", ASME Journal of Heat Transfer, vol. 107, p. 910, 1985.
6. Jambunathan, K., Lai, E., Moss, M. A. and Button, B. L. "A Review Of Heat Transfer Data For Single Circular Jet Impingement", Int. Journal of Heat and Fluid Flow. vol. 13, no. 2, p. 106, 1992.
7. Thomas, B.G., ed., "Mathematical Models of Continuous Casting of Steel Slabs", Annual Report to Continuous Casting Consortium, UIUC, August 19, 1999.

Acknowledgments: The authors wish to thank students Hua Bai, Sivaraj Sivramakrishnan, Shanker Subramanian, and Tiebiao Shi for results referred to in this paper and for help with preparation of figures. Funding from the National Science Foundation (Grant # DMI9800274) and the Continuous Casting Consortium (Allegheny Ludlum Steel, Brackenridge, PA; Armco, Inc., Middletown, OH; Columbus Stainless Steel, Middelburg, South Africa; Inland Steel Co., East Chicago, IN; LTV Steel Co., Cleveland, OH; and Stollberg Inc., Niagara Falls, New York) is gratefully acknowledged. Finally, thanks are extended to the National Center for Supercomputing Applications at the University of Illinois for computing time and use of the CFX code.