© 2014 by Rui Liu. All rights reserved.

MODELING TRANSIENT MULTIPHASE FLOW AND MOLD TOP SURFACE BEHAVIOR IN STEEL CONTINUOUS CASTING

BY

RUI LIU

DISSERTATION

Submitted in partial fulfillment of the requirements for the degree of Doctor of Philosophy in Mechanical Engineering in the Graduate College of the University of Illinois at Urbana-Champaign, 2014

Urbana, Illinois

Doctoral Committee:

Professor Brian G. Thomas, Chair and Director of Research Professor Surya P. Vanka, Contingent Chair Professor Rizwan Uddin Associate Professor Carlos A. Pantano-Rubino

ABSTRACT

This thesis develops, validates and applies a system of computational models to investigate transient multiphase turbulent flow physics in the mold region and mold top surface behavior during continuous casting of steel slabs. Each model can be used independently or combined together as a comprehensive system to gain insights into the inter-related multiphase fluid dynamics in the caster mold during both quasi-steady-state and essentially transient events in practical casting operations.

Argon gas commonly is injected into the liquid metal stream through porous refractory walls in many metallurgical processes. Modeling multiphase flows in caster molds is of great significance to understanding of inclusion transport and defect formation mechanisms and to improving the quality of the final products. To better understand the gas injection process, a new model is developed to investigate gas permeating through heated upper tundish nozzle (UTN) porous refractory, including the effects of nozzle geometry, gas thermal expansion, temperature-dependent gas viscosity, and possible gas leakages into unsealed joints. Furthermore, a procedure to predict initial bubble size is established.

Two (semi-) analytical models, a stopper-position- and a gate-position- based model, predict liquid steel flow rate histories during the transient events and serve as a first step of this comprehensive model system. Argon-steel two-phase flow during a transient "declogging" event with multiple stopper-rod movements is simulated. The flow rate history during stopper rod movements is obtained from the analytical model, and the hot argon flow rate calculated using the porous-flow model and the initial bubble size estimated. Nail board experiments are also conducted to measure steel surface velocities and mold level profiles. A correlation for surface velocity prediction is proposed based on previous modeling results and validated by another set of measurements using a sub-meniscus velocity control (SVC) device.

To further understand particle transport and deposition in wall bounded turbulent flows, direct numerical simulations (DNS) are performed in the continuous phase and a Lagrangian particle tracking algorithm was developed into the in-house code, CU-FLOW, to investigate dispersion and deposition of particles with different Stokes numbers in a square duct flow with and without the effect of imposed magnetic field.

A new free-surface tracking model with a moving-grid technique is developed and integrated in the commercial computational fluid dynamics (CFD) package of ANSYS Fluent (v14.5) based on its dynamic mesh feature, which naturally combines with multiphase flow models. This model is validated and adopted to simulate the dynamic responses of mold top surface to flow rate variations in the SEN subject to the upstream actuator position change.

The complete model system is applied to investigate the effects of slide-gate dithering on transient single- and multi- phase flows in the caster mold. Mold sloshing is identified by both plant experiments and numerical simulations when the dithering frequency matches with the mold natural frequency determined by its geometry. Mechanism for the liquid steel flow variation to activate this standing wave (mold sloshing) is discussed. Multiphase flow pattern and top surface evolution under a low-frequency dithering trial is studied via numerical simulations. Mold level fluctuations are computed from the dithering simulations are compared in favor with the measurements. Up to this point, the model system has been demonstrated a powerful computational tool to resolve complicated multiphase flows during essentially transient events in continuous steel casting subjected to flow rate (both liquid steel and argon gas) variations.

iii

To my parents

ACKNOWLEDGEMENT

I would like to express my deepest gratitude towards my advisers, Prof. Brian G. Thomas and Prof. Pratap S. Vanka, for their continuous guidance, support and encouragement during my entire Ph.D study. This work would not have been possible without their help and patience.

I would like to thank H. Yin, B. Forman, K. Zheng, B. Chukwulebe, K. Blazek, T. Bhattacharya, R. Gass and Y. Yang in ArcelorMittal Global R&D in East Chicago for their help during my internship in the summer of 2011, and for their support in my accomplishing the projects. I would like to show my appreciation towards J. Sengupta, D. Crosbie, S. Chung and M. Trinh for their help during my visit to ArcelorMittal Dofasco in the summer of 2010. I want to give my thanks to R. Nunnington in Magnesita for the enlightening technical discussions and J. Powers and T. Henry for their help with the experiments performed in Severstal at Dearborn, Michigan in Oct, 2012.

Special thanks to the National Center of Supercomputing Applications (NCSA) and especially to Dr. S. Koric and Dr. T. Ahmed for their help in using the Blue Waters clusters. I also would like to give my thanks to the previous and current colleagues of mine in the Metals Processing Simulation Laboratory.

LIST OF FIGURES
LIST OF TABLES
LIST OF SYMBOLS xi
CHAPTER 1: INTRODUCTION
1.1 Continuous Casting of Steel and Fluid Flow-related Phenomena1
1.2 Motivation and Objectives
1.3 Current Work
1.4 Figures11
1.5 References
CHAPTER 2: MODEL OF GAS FLOW THROUGH POROUS REFRACTORYAPPLIED TO
AN UPPER TUNDISH NOZZLE17
2.1 Introduction17
2.2 Mathematical Model Description
2.3 Model Validation
2.4 Model Application Applied to UTN
2.5 Conclusions45
2.6 Tables and Figures48
2.7 References
CHAPTER 3: MEASUREMENTS OF MOLTEN STEEL SURFACE VELOCITYAND
EFFECTS OF STOPPER-ROD MOVEMENT ON TRANSIENT MULTIPHASE
FLUID FLOW IN CONTINUOUS CASTING
3.1 Introduction

TABLE OF CONTENTS

3.2 Surface Velocity Measurement in Mol	ten Steel Caster66
3.3 Plant Measurements	
3.4 Validation and Calibration of Nail-Dip	pping Method70
3.5 Computational Model Description	
3.6 Investigation of Casting Condition Eff	ects on Surface Velocity76
3.7 Transient Event Simulation Results	
3.8 Summary	
3.9 Figures and Tables	
3.10 References	
CHAPTER 4: PARTICLE TRANSPORT AND	DEPOSITION IN A TURBULENT SQUARE
DUCT FLOW WITH AN IMPOS	SED MAGNETIC FIELD
4.1 Introduction	
4.2 Numerical Formulation	
4.3 Results and Discussion	
4.3.1 Continuous Flow Fields	
4.3.2 Particle Dispersion	
4.3.3 Particle Deposition	
4.4 Summary	
4.5 Figures and Tables	
4.6 References	
CHAPTER 5: COMPUTATIONAL AND EXP	ERIMENTAL STUDIES OF ARGON-STEEL
TWO-PHASE FLOWS IN A CO	NTINUOUS CASTER MOLD142
5.1 Introduction	

	5.2 Plant Trials with Nail-Board Measurements	145
	5.3 Model Description and Computation Details	147
	5.4 Results and Discussion	158
	5.5 Summary	165
	5.6 Figures and Tables	167
	5.7 References	178
CHA	PTER 6: MODELING TRANSIENT MULTIPHASE FLOW AND TOP SURFACE	
	DYNAMICS IN A CONTINUOUS CASTER MOLD DURING SLIDE-GAT	Е
	DITHERING	184
	6.1 Introduction	184
	6.2 Plant Measurements	189
	6.3 Model Description	191
	6.3.1 Gate-Position-based Flow Rate Model	192
	6.3.2 Gas Porous-Flow Model	194
	6.3.3 3-D Transient Eulerian Multiphase Model for SEN/Mold Flow	196
	6.3.4 Free-Surface Tracking Algorithm using Moving-Grid Technique	200
	6.4 Model Validation	204
	6.5 Quasi-steady Results with Single-Phase Flow	208
	6.6 Dithering Results with Single-Phase Flows	209
	6.7 Dithering Results with Multiphase Flows	215
	6.8 Summary	224
	6.9 Figures and Tables	226
	6.10 References	244

CHAPTER 7: CONCLUSIONS		
		7 259
CHAPTER 8: RECOMMENDAT	IONS FOR FUIURE WORI	Δ

PRESSURE-SOURCE MODEL IN CYLINDRICAL COORDINATE

10
[
)3

APPENDIX E. A GENERALIZED DERIVATION OF MASS AND MOMENTUM SINKS AT

SHELL INTERFACE)4
-----------------	----

LIST OF FIGURES

Figure 1.1: Schematic of continuous casting process
Figure 1.2: Schematic of caster nozzle/mold region, (a) flow control system; (b) complex
transport phenomena in mold region11
Figure 1.3: Particle types and morphologies, (a) Alumina particle; (b) Alumina cluster; (c) Slag
droplet inclusions; (d) Argon bubble (contaminated with particles)12
Figure 2.1: Schematic of upper tundish nozzle (UTN) in continuous casting process
Figure 2.2: Bubble formation stages at refractory surface showing expansion and pressure
threshold (Stage 2)
Figure 2.3: Geometry, mesh and boundary conditions for 1-D test cases
Figure 2.4: Temperature and argon viscosity profiles in radial direction
Figure 2.5: Pressure distributions for 1-D test cases with fixed pressure B.C
Figure 2.6: Pressure distributions for 1-D test cases with fixed gas mass flow rate B.C
Figure 2.7: Comparison between predicted and measured gas flow rate at UTN inner surface, (a)
Photograph of underwater experiment showing bubbling locations; (b) Predicted gas
normal surface velocity distribution
Figure 2.8: Gas velocity distribution at symmetry plane, (a) Contours of velocity magnitude; (b)
Velocity vectors
Figure 2.9: Schematic of the base case UTN geometry (dimensions in mm)

Figure 2.10: Computational domain, mesh and boundary condition
Figure 2.11: Temperature distribution across UTN wall
Figure 2.12: Effect of liquid steel pressure profile on gauge pressure distribution, (a) Case 1:
linear pressure with bubbling threshold; (b) Case 2: linear pressure (Fig.2.10); (c)
Case 3: constant pressure54
Figure 2.13: Radial velocity distribution at UTN inner surface
Figure 2.14: Velocity near UTN bottom showing effect of boundary conditions, (a) With one-
way flow pressure B.C.; (b) With fixed pressure B.C
Figure 2.15: Pressure (gauge) and velocity fields for, (a) Open bottom case; (b) Perfectly sealed
case
Figure 2.16: Effect of gas leakage on radial gas velocity distribution at UTN inner surface57
Figure 2.17: Effect of injection pressure (gauge) on gas velocity distribution in UTN refractory
(sealed bottom), (a) 90 kPa; (b) 99 kPa; (c) 140 kPa57
Figure 2.18: Effects of injection pressure (gauge) and bottom sealing on gas velocity distribution
at UTN inner surface
Figure 2.19: Effects of injection pressure (gauge) on gas flow rate
Figure 2.20: Effects of permeability on radial velocity distribution
Figure 2.21: Effects of permeability on gas mass flow rate
Figure 3.1: Nail dipping procedure, (a) Surface slag layers; (b) Nail and lump

Figure 3.2: SEM image of subsurface defects in trial #3	87
Figure 3.3: Curves to convert nail lump height difference into velocity magnitude at the top	
surface	88
Figure 3.4: Casting speed change in trial #1 and measured meniscus velocity, (a) casting spee	ed
change; (b) meniscus velocity history from SVC and nail dipping	89
Figure 3.5: Effects of C_2 and C_3 on stopper-position-based model predictions of flow rate	89
Figure 3.6: Computational domain for full mold simulation, (a) geometry and (b) cutaway vie	ew
of mesh	90
Figure 3.7: Effects of casting speed, mold width and gas volume fraction (in hot condition) or	n
surface velocity (trials #1 and #2)	90
Figure 3.8: Flow patterns with different argon gas volume fractions, with varying casting	
speeds	91
Figure 3.9: Comparison between predicted and measured surface velocities from trial #2	91
Figure 3.10: Quasi-steady state argon volume fraction distribution (trial #3)	92
Figure 3.11: Recorded stopper rod position, mold level and casting speed (trial #3)	92
Figure 3.12: Predicted SEN flow rates from stopper-position-based and metal-level-based mo	odel
(for trial #3)	93
Figure 3.13: Transient flow patterns during stopper rod movement in trial #3	
(axis distances in m)	94

Figure 3.14: Comparison of calculated and measured mold level (for trail #3)95
Figure 4.1: Schematic of computational domain
Figure 4.2: Velocity profiles along horizontal and vertical bisectors in MHD duct flow125
Figure 4.3: Instantaneous velocity field in a cross-sectional plane, (a) Non-MHD case; (b) MHD
case
Figure 4.4: Time-averaged velocity field in a cross-sectional plane, (a) Non-MHD case; (b)
MHD case
Figure 4.5: Instantaneous particle positions in a cross-sectional plane, (a) without MHD, (St=5);
(b) with MHD, (St=5); (c) without MHD, (St=15); (d) with MHD, (St=15)128
Figure 4.6: Preferential particle concentration in a cross-sectional plane, (a) Non-MHD case,
St=5; (b) MHD case, St=5; (c) Non-MHD case, St=15; (d) MHD case, St=15129
Figure 4.7: Instantaneous streamwise velocity contours with particle positions for St=5, (a) Non-
MHD case; (b) With MHD case, at $z^+=5$; (c) With MHD case, at $y^+=5$ 130
Figure 4.8: Instantaneous streamwise velocity contours with particle positions for St=15, (a)
Non-MHD case; (b) With MHD case, at $z^+=5$; (c) With MHD case, at $y^+=5$ 131
Figure 4.9: Probability distribution function of particle deposition location along streamwise
direction132
Figure 4.10: Probability distribution function of particle deposition location along spanwise and
transverse directions, (a) pdf on walls parallel to magnetic field; (b) pdf on walls
perpendicular to magnetic field

Figure 4.11: Comparison of particle deposition pdf for two particle releasing approaches134
Figure 4.12: Particle deposition locations on walls parallel to the magnetic field (St=5), (a) Non-
MHD case; (b) MHD case
Figure 4.13: Particle deposition locations on walls perpendicular to the magnetic field (St=5), (a)
Non-MHD case; (b) MHD case135
Figure 4.14: Wall-normal velocity distribution of depositing particles, (a) Non-MHD case; (b)
MHD case, on walls perpendicular to magnetic field; (c) MHD case, on walls
parallel to magnetic field136
Figure 4.15: Streamwise velocity distribution of depositing particles, (a) Non-MHD case; (b)
MHD case, on walls perpendicular to magnetic field; (c) MHD case, on walls
parallel to magnetic field137
Figure 4.16: Dimensionless particle deposition rates
Figure 5.1: Schematic of transport phenomena in continuous casting mold168
Figure 5.2: Photo of the dipped nail boards
Figure 5.3: Flow chart for initial bubble size prediction procedure
Figure 5.4: Schematic of UTN domain and mesh169
Figure 5.5: Domain and mesh for mold flow simulations170
Figure 5.6: UTN temperature field170
Figure 5.7: Gas velocity distribution in UTN refractory171

Figure 5.8: Initial bubble size distribution
Figure 5.9: Single phase steel flow pattern at mold center plane (DES results), (a) Instantaneous
velocity field, Case 1; (b) Mean velocity field, Case 1; (c) Instantaneous velocity
field, Case 2; (d) Mean velocity field, Case 2; (e) Instantaneous velocity field, Case
3; (f) Mean velocity field, Case 3;172
Figure 5.10: Single phase flow results (URANS model)173
Figure 5.11: Two-phase flow field at mold center plane, (a) Liquid steel velocity distribution; (b)
Gas velocity distribution173
Figure 5.12: Mold top surface velocity profiles – Case 1174
Figure 5.13: Mold level profile – Case 1174
Figure 5.14: Bubble size effect on liquid steel flow patterns – Case 2, (a) 3mm bubble diameter;
(b) 5 mm bubble diameter174
Figure 5.15: Bubble size effect on argon gas flow patterns – Case 2, (a) 3 mm bubble diameter;
(b) 5 mm bubble diameter175
Figure 5.16: Mold top surface shape from moving-grid surface tracking model results175
Figure 5.17: Top surface steel velocity profiles – Case 2175
Figure 5.18: Mold top level profile – Case 2176
Figure 5.19: Center plane velocity distribution – Case 3 for, (a) Liquid steel velocity; (b) argon
gas velocity and volume fraction176

Figure 5.20: Top surface steel velocity Distribution – Case 3 for, (a) Liquid steel velocity; (b)
argon gas velocity and volume fraction176
Figure 5.21: Top surface velocity profiles – Case 3
Figure 5.22: Mold level profile – Case 3
Figure 6.1: Dithering trials with mold level measurements
Figure 6.2: Influence of dithering frequency on mold level fluctuations
Figure 6.3: Schematic of sloshing mode indices
Figure 6.4: Mold natural frequency curves
Figure 6.5: Model system for dithering process simulations
Figure 6.6: Top view of gate and SEN inner bores
Figure 6.7: Domain geometry and mesh setup, (a) Domain geometry; (b) Zoomed-in region A;
(c) Zoomed-in region B
Figure 6.8: 3-D Boundary surface mesh layout and surface tension calculation
Figure 6.9: Comparison of flow rate model predictions with water model measurements229
Figure 6.10: Comparison between measured and AMLE-predicted mold level fluctuations230
Figure 6.11: Schematic for small-amplitude sloshing in a tank
Figure 6.12: Validation of the free-surface tracking model

Figure 6.13: Comparison of simulated mold center plane flow patterns by different turbulence
models, (a) By k - ω model; (b) By DES model
Figure 6.14: Comparison of simulated mold top surface morphologies by different turbulence
models, (a) By k - ω model; (b) By DES model
Figure 6.15: Zoomed-in of free surface close to SEN calculated using DES (length magnified by
5 times along z-axis), (a) Free surface morphology with mesh deformation; (b) Top
surface liquid steel velocity close to SEN231
Figure 6.16: Transient liquid steel flow pattern evolution during one dithering cycle for the mold
sloshing Case 1
Figure 6.17: Mold top surface behavior during one dithering cycle (Case 1)
Figure 6.18: Full mold flow evolution during a dithering cycle with mold sloshing (Case 1)234
Figure 6.19: Comparison between measured and calculated mold level at quarter point
(Case 1)234
Figure 6.20: Mold level evolution during a typical dithering cycle (Case 1)
Figure 6.21: Maximum sloshing amplitude over 8 dithering cycles (Case 1)235
Figure 6.22: Flow pattern evolution at dithering frequency 1.4 Hz (Case 2)
Figure 6.23: Mold top surface evolution (Case 2)
Figure 6.24: Effects of dithering stroke, casting speed and mold width on flow rate variation and
mold level fluctuation ($f=1.0$ Hz), (a) Correlation between flow rate variation and

stroke; (b) Mold level fluctuation dependence; (c) Effects of casting speed and mold
width
Figure 6.25: Slide-gate position history and measured mold level
Figure 6.26: Computed histories (<i>f</i> =0.4 Hz) of liquid steel flow rate, (a), argon flow rate and
volume fraction (b) and initial bubble size (c)
Figure 6.27: Initial bubble size distribution during slide-gate dithering
Figure 6.28: Initial flow fields for multiphase dithering case, (a) Liquid steel velocity
distribution; (b) Argon distribution239
Figure 6.29: Initial solution of steel velocity and argon fraction at mold top surface239
Figure 6.30: Liquid steel velocity distribution at SEN port exit, axis in (m)240
Figure 6.31: Argon velocity and concentration distribution at SEN port exit, axis in (m)240
Figure 6.32: Evolving of liquid steel velocity distribution at center plane during dithering,
axis in (m)241
Figure 6.33: Argon gas fraction variation during dithering process, showing periodic gas pockets,
axis in (m)242
Figure 6.34: Measured and predicted mold level histories using both the pressure method and the
moving mesh model (m)242
Figure 6.35: Steel flow patter evolution during one dithering cycle (<i>f</i> =0.4 Hz)243
Figure 6.26: Argon volume fraction evolution during one dithering cycle (<i>f</i> =0.4 Hz)243

Figure 6.37: Liquid steel flow rate and mold level signals
Figure A.1: Domain sketch and linear pressure distribution assumption
Figure A.2: Velocity profiles across 1 mm mold-shell gap at different shell temperatures269
Figure A.3: Comparison of analytical solution with numerical simulation
Figure A.4: Velocity profiles across 3 mm mold-shell gap at different shell temperatures –
analytical and numerical solutions
Figure A.5: Sketch of the domain for analysis
Figure A.6: Shell temperature (linear variation)
Figure A.7: Pressure distribution down the mold (Scenario I linear distribution vs.
calculated)
Figure A.8: Pressure gradient down the mold (Scenario I linear distribution vs. calculated)282
Figure A.9: Downward velocity profile across the gap (Scenario II)
Figure A.10: Comparison of velocity profiles from Scenario I and current model
Figure A.11: Temperature distribution along casting direction
Figure A.12: Liquid flux layer thickness vs. distance down the mold (wide face)
Figure A.13: Comparison of pressure from different boundary conditions
Figure A.14: Vertical velocity profile for B.C. type 1 (ferrostatic pressure at gap outlet)
Figure A.15: Vertical velocity profile for B.C. type 2 (1 atm at gap outlet)

Figure A.16: Liquid flux layer thickness vs. distance down the mold (narrow face)290
Figure A.17: Comparison of pressure from different boundary conditions
Figure A.18: Velocity profile for B.C. type 1 (ferrostatic pressure at gap outlet)
Figure A.19: Velocity profile across liquid flux layer (1 atm at the gap outlet at
mold bottom)
Figure A.20: Liquid flux layer thickness vs. distance down the mold (mold corner)292
Figure A.21: Comparison of pressure profile from different boundary conditions293
Figure A.22: Downward velocity profile across the liquid flux layer (B.C. type 1)
Figure A.23: Downward velocity profile across the liquid flux layer (B.C. type 2)
Figure A.24: Pressure distribution for different gaps
Figure A.25: Vertical velocity profile for different gaps at mold exit
Figure E.1: Control volume adjacent to shell interface

LIST OF TABLES

Table 2.1: Parameters for 1-D test case boundary conditions (based on absolute pressure)48
Table 2.2: Parameters for upper tundish nozzle (UTN) heat transfer analysis
Table 2.3: Refractory properties and casting conditions in sarametric study (base case)
Table 3.1: Processing Parameters for Plant Trials 86
Table 4.1: Particle properties used in the simulation 124
Table 5.1: Casting conditions for nail board trials
Table 5.2: Upper tundish nozzle (UTN) refractory properties
Table 6.1: Casting conditions for dithering trials 226
Table A.1: Case parameters (Scenario I.) 267
Table A.2: Numerical integration parameters and constants 287

LIST OF SYMBOLS

V	Velocity (gas, particle, liquid steel, mixture phase)
u	Velocity of liquid steel
р	Pressure
Т	Temperature
f	Dithering frequency
Q	Volumetric flow rate
K_D	Permeability
K_{DS}	Specific permeability
k	Thermal conductivity
μ	Dynamic viscosity of fluid
Κ	Thermal conductivity
ρ	Density
D	Diameter
n	Unit surface normal vector
k	Turbulent energy
κ	Surface curvature
σ	Surface tension
r	Radial position
r _{pore}	Refractory pore size

Α	Area vector
V _C	Casting speed
V_S	Molten steel surface velocity
ϕ_{lump}	Lump diameter
h _{lump}	Lump height difference
α	Volume fraction
В	Magnetic field
$ au_p$	(Particle) relaxation time
g	Gravity
F _{drag}	Drag force
C_D	Drag coefficient
$\mathbf{F}_{\mathrm{lift}}$	Lift force
F _{added-mass}	Added mass force
F _{press_grad}	Pressure gradient force
F _B	Buoyancy force
\mathbf{F}_{G}	Gravity force
g	Gravity accerlation
x	Position vector
Т	Time
Δh	Level elevation
J	Current
Re	Reynolds number
Pr	Prandtl number

Nu Nusselt number

St Particle Stokes number

CHAPTER 1. INTRODUCTION

Continuous casting has been a predominant way of steel production world-wide over decades, which contributes more than 95% of the 1.5 billion tonnes of steel produced every year. Many defects and quality problems are caused by turbulent fluid flow in the mold – especially at the top surface. This mature process is difficult to improve without better quantitative understanding. However, very few flow-related quantities can be measured in the real molten steel, so mathematical modeling becomes a necessary tool to reveal the flow-related phenomena. Due to the complex, interrelated nature of the transient, multiphase, turbulent flow in this process and limitations in the measurements, accurate flow model predictions are difficult.

The first section briefly introduces the continuous casting process with an emphasis on the flow control methods and the formation of flow-related defects. The second section presents uncertainties that exist in the quantitative study of this complex process but are not well understood. The last section provides an overview of this thesis, which aims to improve the accuracy of computational models to quantify mold flow behavior, by establishing a comprehensive model system to investigate complex transient behavior of argon-steel two-phase flows and mold top surface behavior during both essentially transient and quasi-steady process conditions.

1.1. Continuous Casting of Steel and Fluid Flow-related Phenomena

As illustrated in Figure 1.1,^[1] during the continuous casting process, molten steel is continuously drained from a ladle through a shroud into a tundish. Liquid steel exits the tundish and flows downward through an upper tundish nozzle (UTN) at the tundish bottom and a

submerged entry nozzle (SEN) driven by gravity. The liquid steel flow rate is controlled by adjusting the actuator positions in the flow control devices, which consists of either a stopper-rod system or a slide-gate system (Figure $1.2a^{[2]}$) For the stopper-rod system, molten steel flows through the gap area between the stopper-rod tip and the UTN, where pressure drop occurs to achieve the desired flow rate. In the slide-gate system, the position of a moving plate with a central hole (slide gate) determines the alignment and size of the gate opening, which thus controls the pressure drop and thus the flow rate.

Further down the SEN, liquid steel enters the caster mold from bifurcated SEN ports, where the liquid steel flows as turbulent jets across the mold cavity towards the narrow faces of the slab mold. The steel starts to solidify near the water-cooled mold walls to form a solid shell, as shown by the schematic in Figure 1.2.^[2-3] Impurity particles, such as alumina, travel downward in the steel stream from upstream into the liquid pool in the mold region. Continuous deposition of these inclusion particles onto the nozzle inner surface leads to clogging formation and reduction of the effective flow area in the nozzle, which in turn increases the pressure drop and reduces liquid steel flow rate in the SEN. The inclusion particles could also travel near the shell interface, get entrapped into the solidifying shell and become defects in the final product. This steel shell keeps growing as it is withdrawn downward in the casting direction, so that at mold exit it is strong enough to endure the ferrostatic pressure of the liquid steel pool and avoid breakouts. As the strand travels further, the liquid steel contained in the shell continuously solidifies through the secondary cooling zone, and after becoming fully solid, is finally torch cut off into slabs for rolling into flat products. (Figure 1.1).

As shown in Figure 1.2(a),^[2] argon is injected into the UTN through a series of internal slits, and permeates through the porous portions of the refractory wall into the liquid steel stream.

The gas emerges from the UTN inner surface, and is usually sheared off into bubbles by the downward-flowing liquid steel stream. At high flow rates, this injected gas might form a gas curtain on the UTN/SEN inner surfaces to prevent the molten steel from touching the refractory wall, where it could solidify and exacerbate clogging of the nozzle. The gas also creates an argon atmosphere and pressure both within the refractory and inside the nozzle that helps to prevent air aspiration and consequent re-oxidation of liquid steel.

As illustrated in Figure 1.2(b),^[3] slag powder is distributed on top of the molten steel surface in the mold, and melts into a layer of liquid flux that insulates the steel from both heat loss to the ambient atmosphere and from re-oxidation of the metal. Another benefit from the injected argon bubbles is that their surfaces gather small inclusion particles via collisions during their travel through the liquid steel, and usually carry them up to the liquid flux layer where the particles may be absorbed into the slag layer and improve cleanliness of the liquid steel. Another function of the mold slag is that the molten flux fills the mold-shell gap during the mold oscillations and flows downward along the gap, which consumes the liquid flux and controls heat transfer across the interface from the steel shell to the mold wall.^[4-5] Thus, the flux controls uniformity of the heat transfer across the gap, which is important to avoid the formation of longitudinal cracks. The liquid flux pressure and velocity profiles across the mold-shell gap with varying gap size and temperature-dependent flux viscosity are resolved analytically and presented in Appendix A. The shape of the top surface of the molten steel / slag interface is important to enable uniform slag consumption into the gap. Cracks and surface defects may also form if the meniscus becomes too cold, due to insufficient surface flow. Slag droplets could get entrained into the liquid steel pool via many mechanisms,^[6-7] leading to internal inclusion defects. Slag entrainment has recently been summarized into 9 mechanisms.^[8] For instance, slag droplets

get sheared off from the flux layer under certain flow conditions when the steel surface velocity exceeds a critical value.^[6] Intensive mold level fluctuations at the meniscus around the top surface perimeter may also lead to slag entrainment into the liquid steel pool.^[7]

1.2. Motivation and Objectives

Complex transport phenomena occur due to turbulent flow of molten steel, inclusion particles and slag droplets in the mold region, especially when argon gas bubbles are injected, as shown in Figure 1.2(b). Many surface defects in the final steel products (e.g. slivers, blisters) are caused by the entrapment of argon bubbles, slag droplets and other particles (e.g. alumina particles or clusters) into the solidifying shell. The morphologies of some typical entrapped inclusions are shown in Figure 1.2(a)-(d), including the alumina particle or cluster,^[11] slag droplets,^[12] and contaminated argon bubbles.^[13]

Flow behavior near the flux-steel interface is crucial to product quality and affects defect formation in multiple ways. Liquid steel surface velocity (at flux-steel interface) is an important indicator of flow-related problems: too high a surface velocity leads to excessive turbulence and shear instability at slag-steel interface and increases the possibility of slag entrainment; too low a surface velocity results in too much cooling near the meniscus regions, which may further cause hook formation, nonuniform slag consumption, and the entrapment of mold slag, inclusion particles, or bubbles, leading to various surface defects.^[19] Excessive mold level fluctuations also cause slag entrainment. Most of these problems are intermittent, due to the transient nature of turbulent flow. Thus it is of great importance to find, validate, and apply methods to quantify surface velocity in the mold, including transient behavior.

As a basis to quantitatively understand the flow behavior near flux-steel interface, particle transport and entrapment,, turbulent flows in the mold region should be accurately calculated a priori. Liquid steel flow patterns in continuous caster molds have been intensively studied over decades using both scaled mockup "cold" experiments and numerical simulations under different operation conditions, which have been well summarized and discussed by Thomas and Zhang.^[15] Most of the previous studies focused on modeling (quasi-) steady state single-phase liquid steel flows or argon-steel two phase flows with known gas injection rates.

Defects often form during essentially transient processes (e.g. sudden or periodic variations of actuator positions) with flow pattern changes, which have not received much attention in previous research, partly due to the practical difficulties in obtaining the quantitative variations during these events as input conditions for transient simulation. For instance, for the transient events related to liquid steel flow rate change in the system, movements of the flow control devices, either a stopper-rod or a slide-gate system, cause the flow rate in the SEN to vary, and this variation cannot be measured. Mathematical models become a necessary tool to predict this flow rate change before further investigation is possible.

In addition, uncertainties rise in the practical operations during gas injection regarding two key parameters: argon flow rate entering liquid steel stream and the initial bubble size, which further complicate the investigation of these transient processes. The volumetric flow rate of the injected argon gas is usually measured in the "cold", standard temperature and pressure (STP) condition well before entering the nozzle, in standard liters per minute (SLPM). This is usually much smaller than the flow rate entering the molten steel in hot condition upon gas leaving the UTN inner surface, due to gas thermal expansion. In previous work, this effect is accounted for with the ideal gas law, considering local temperature and estimated pressure.^[16] This simple treatment works reasonably well under the assumptions of zero gas leakage uniform pressure/temperature distributions at both the gas injection and exit surfaces, which are not usually satisfied in the practical applications. Some of the gas escapes through leaks in the delivery system, cracks in the refractory or imperfect seals at the joints between refractory components. The remaining gas expands as it travels through the heated refractories and enters the steel flow with a non-uniform distribution. The initial bubble size also depends on this gas flow rate, as well as the cross-flow liquid (steel) velocity and material properties of argon, liquid steel and the UTN refractory. Thus a more accurate model is required to study the gas flow through porous UTN refractory and to predict the initial bubble sizes.

For model validation purposes, cold physical model (e.g. water model) experiments are intensively performed to measure velocities and surface level profiles, using laser Doppler anemometry (LDA)^[17] or particle image velocimetry (PIV) techniques.^[18] These laboratory measurements could help with understanding the flow dynamics in caster molds via similarity principles, and to validate computational models. For flows with gas injection, as pointed out by Huang and Thomas,^[11] multi-phase flow behavior in an air-water system differs from that in the corresponding argon-steel system in many ways, due to differences in properties such as surface tension and contact angles leading to different bubble sizes. Similarly, oil-water systems are essentially different than molten slag-steel systems. Thus, plant measurements are preferred over water models, especially for multiphase flows.

However, steel flows in the mold cannot be directly observed due to the opaque mold walls and the sintered slag layer on top of the molten metal, and very limited quantities regarding liquid steel flow can be visualized or measured. Molten steel surface velocity can be measured via different techniques invented during past decades to study mold fluid flow in operating casters,^[19-21] among which the nail-board approach is recently gaining popularity due to its simplicity in application and ability to measure both the direction and magnitude of the surface instantaneous velocities. Previous work^[22-24] has successfully applied this method to measure surface velocities for model validation purposes, but the quantification of the steel surface velocities from the dipped nails has not yet been validated by experiments.

1.3. Current Work

Considering the complexity of fluid flows in the continuous casting process and the difficulties embedded in both plant measurements and numerical simulations, numerical tools have been developed to gain a deeper understanding of flow behavior during transient events with argon injections. In this thesis, a comprehensive model system is presented, which consists of several different models. Each model is explained in a separate chapter, validated with analytical solutions of the test problem and experiment measurements, and applied to gain new insight into specific processes during continuous casting. Finally, the complete model system is applied to study complex transient two-phase flows in the mold region during slide-gate dithering, with a novel free-surface tracking model to predict mold top surface motion.

Chapter 2 introduces a porous-flow model and a pressure-source model, which have been developed to simulate gas flow inside the porous refractory, and bubble formation. These models compute the gas superficial velocity distribution that enters liquid steel. The models can also predict possible gas leakage during injection by comparing the simulated gas mass flow rate with the measurements. These models (porous-flow and pressure-source models) thus provide valuable insights into the gas injection process, and eliminate the uncertainties in measuring and adopting the correct gas flow rates in the subsequent argon-steel flow simulations. In addition to obtaining the gas velocities (flow rates), a procedure is established in this work to estimate the initial bubble sizes combining a previous model (Bai, 2001)^[25] and correlation (Lee, 2010).^[26] Both the calculated argon flow rate and the estimated initial bubble size are used as input conditions for argon-steel two-phase flow simulations. The UTN used in the bubbling experiment was provided by J. Sengupta in ArcelorMittal Dofasco, and B. Forman in ArcelorMittal Global R&D in East Chicago helped with the experiment setup.

Chapter 3 proposes a correlation to quantify liquid steel surface velocities using the solidified lump height difference around the nail perimeter, based on a least-square regression of previous simulation results.^[24] This correlation is then validated by the velocity signals measured simultaneously at (approximately) the same location using a sub-meniscus velocity control (SVC) device. A semi-analytical model predicting liquid steel flow rate change based on the measured stopper-rod position history was firstly developed and validated with plant measurements, and then applied to calculate the inlet liquid steel flow rate history as an input to the multiphase flow simulations. The developed models are applied to study a transient event where multiple stopper-rod movements occurred and caused defects formation identified by a downstream quality monitoring system.

The plant trial with nail-dipping and SVC experiments in this chapter were coordinated by J. Sengupta, S. Chung and M. Trinh in ArcelorMittal Dofasco. J. Sengupta also helped with the surface velocity measurements on the dipped nails.

Chapter 4 develops a multiphase flow model with direct numerical simulations (DNS) for the continuous fluid phase, and a Lagrangian particle tracking algorithm for the dispersing particles. This multiphase model is integrated into an in-house GPU-based code, CU-FLOW, to study particle dispersion and deposition in a simple test domain: turbulent flow in a square duct

with a frictional Reynolds number of 360 (Re_{τ} =360). Particles with several Stokes numbers (0.1-15) were simulated with one-way coupling to study the effect of particle diameter on the deposition rate and pattern. The effect of an imposed magnetic field on the turbulent flow and the particle dynamics is also investigated.

Chapter 5 develops and evaluates two sets of multiphase flow models, with both the Eulerian-Eulerian and the Eulerian-Lagrangian model. The porous-flow model in Chapter 2 is first used to calculate the gas velocity distributions in the nozzle refractory, and the initial bubble sizes are predicted for the simulations. Two different surface level tracking models, a simple pressure method and a moving-grid model, are applied to predict mold levels during the trial. The simulated results with surface velocities and level positions are then compared with nailboard measurements with reasonable matches achieved. Effect of bubble size in the resulting flow patterns is also discussed. The plant trials for the nail-board experiments were carried out in Severstal, Dearborn in Michigan and coordinated by J. Powers and T. Henry.

Chapter 6 applies the complete model system to investigate flow pattern evolution and mold top surface motion subject to slide-gate dithering with and without argon gas injection. To predict the time-varying liquid steel flow rate rate in the nozzle due to slide gate oscillation, a gate-position-based model is derived analytically based on the Bernoulli's equation. Next, the argon flow rate entering the liquid steel during the slide gate dithering and the corresponding initial bubble size are predicted using the model presented in Chapter 5, but with varying liquid steel cross-flow velocities. This liquid steel flow rate change leads to variations in argon flow rate as well as in the initial bubble size. To model the dynamic response of the mold top surface to the multiphase flow variations, a novel free-surface tracking algorithm using a moving-grid technique is developed, which includes the effect of slag-steel surface tension at the interface.

This surface-tracking methodology is validated against the analytical solution for a smallamplitude tank sloshing problem.^[24] Finally, the Eulerian-Eulerian multiphase model is applied to simulate the flow pattern evolutions during gate dithering, with the previously calculated liquid steel and argon flow rate histories and initial bubble size as inputs, and using the new free surface method. The dithering trials were conducted in Indiana Harbor 3SP by A. Dasgupta, L. Kalra, H. Yin, T. Bhattacharya, K. Zheng and B. Umlauf in ArcelorMittal Global R&D in East Chicago.

Chapter 7 summarizes the major findings in each of the chapters of this work, including both model development and respective applications to increase understanding of the continuous casting process. (Appendix B lists the publications regarding the current work)

Chapter 8 suggests future work regarding future improvement and application of the current model system to study complex transient multiphase flow / multiphysics phenomena in the caster mold.

1.4. Figures



Figure 1.1. Schematic of continuous casting process^[1]



Figure 1.2. Schematic of caster nozzle / mold region (a) flow control system^[2] (b) complex transport phenomena^[3]


(c) Slag droplet inclusions^[13]



(b) Alumina cluster^[12]



(d) Argon bubble^[14]

Figure 1.3. Particle types and morphologies

1.4. References:

- [1] X. Huang and B.G. Thomas, "Modeling of Steel Grade Transition in Continuous Slab Casting Processes", *Metallurgical and Materials Transaction B*, 24(2), 1993, pp. 379-393.
- [2] D. Hershey, B.G. Thomas and F.M. Najjar, "Turbulent Flow through Bifurcated Nozzles", *International Journal for Numerical Methods in Fluids*, 1993, vol. 17 (1), pp. 23-47.
- Y. Meng and B.G. Thomas, "Heat-Transfer and Solidification Model of Continuous Slab Casting: CON1D", *Metallurgical and Materials Transactions B*, 34B(5), 2003, pp. 685-705.
- [4] R. M. McDavid and B. G. Thomas, "Flow and Thermal Behavior of the Top Surface Flux/Powder Layers in Continuous Casting Molds", *Metallurgical and Materials Transactions B*, 27, 1996, pp. 672-685.
- [5] P.E. Ramirez-Lopez, P.D. Lee and K.C. Mills, "Explicit Modelling of Slag Infiltration and Shell Formation During Mould Oscillation in Continuous Casting", *ISIJ International*, 2010, 50, pp. 425-434.
- [6] T. Watanabe and M. Iguchi, "Water model experiments on the effect of an argon bubble on the meniscus near the immersion nozzle", *ISIJ International*, 2009, 49(2) pp. 182–188.
- B.G. Thomas, "Modeling of Continuous Casting Defects Related to Mold Fluid Flow", 3rd Internat. Congress on Science & Technology of Steelmaking, Charlotte, NC, May 9-12, AIST, Warrendale, PA, 2005, pp. 847-861.
- [8] L.C. Hibbeler and B.G. Thomas, "Mold Slag Entrainment Mechanisms in Continuous Casting Molds", *AISTech 2013 Proceedings*, Pittsburgh, PA, May 6-8, 2013, Assoc. Iron Steel Technology, Warrendale, PA, pp. 1215-1230.

- [9] Q. Yuan, B.G. Thomas, and S.P. Vanka, "Study of Transient Flow and Particle Transport during Continuous Casting of Steel Slabs, Part 2. Particle Transport", *Metallurgical and Materials Transactions B*, 2004, 35B(4), pp. 703-714.
- [10] B.G. Thomas, Q. Yuan, S. Mahmood, R. Liu and R. Chaudhary, "Transport and Entrapment of Particles in Steel Continuous Casting", *Metallurgical and Materials Transactions B*, 2014, 45(1), pp. 22-35.
- [11] R.A. Rege, E.S. Szekeres, and W.D. Forgeng, "Three Dimensional View of Alumina Clusters in Al-killed Low Carbon Steel", *Metallurgical and Materials Transactions B*, AIME, 1970, 1, p. 2652.
- [12] L. Zhang and B.G. Thomas, "State of Art in Evaluation and Control of Steel Cleanliness", *ISIJ International*, 2003, 43(3), pp. 271–291.
- [13] Y. Miki and S. Takeuchi, "Internal Defects of Continuous Casting Slabs Caused by Asymmetric Unbalanced Steel Flow in Mold", *ISIJ International*, 2003, 43(10), pp. 1548–1555.
- [14] J.A. Dantzig and M. Rappaz, "Solidification", *EPFL Press*, 2009, pp. 323.
- [15] B.G. Thomas and L. Zhang, "Mathematical Modeling of Fluid Flow in Continuous Casting", *International Steel and Iron Journal*, 41(10), 2001, pp. 1181-1193.
- [16] X. Huang and B.G. Thomas: "Modeling of Transient Flow Phenomena in Continuous Casting of Steel", *Canadian Metallurgical Quarterly*, 1998, vol. 37(304), pp. 197-212.
- [17] N.J. Lawson and M.R. Davidson, "Oscillatory Flow in a Physical Model of a Thin Slab Casting Mould with a Bifurcated Submerged Entry Nozzle", *Journal of Fluids Engineering*, 2002, 124(6), pp. 535-543.

- [18] R.J. Adrian, "Particle-Imaging Techniques for Experimental Fluid Mechanics", Annual Review of Fluid Mechanics, 1991, 23, pp. 261-304.
- [19] P.H. Dauby, W.H. Emling, and R. Sobolewski, "Lubrication in the Mold: A Multiple Variable System", *Ironmaker and Steelmaker*, 1986, 13, p. 28-36.
- [20] M. Iguchi, et al., "A New Probe for Directly Measuring Flow Velocity in a Continuous Casting Mold", *ISIJ International*, 1996, 36(Suppl.), pp. S190-S193.
- [21] B. Melissari and S.A. Argyropoulos, "Measurement of Magnitude and Direction of Velocity in High-Temperature Liquid Metals. Part II: Experimental measurements", *Metallurgical and Materials Transaction B*, 2005, 36(5), pp. 639-649.
- [22] R. McDavid and B.G. Thomas, "Flow and Thermal Behavior of the Top-Surface Flux/ Powder Layers in Continuous Casting Molds", *Metallurgical and Materials Transaction B*, 1996, 27B(4), pp. 672-685.
- [23] K. Cukierski and B.G. Thomas, "Flow Control with Local Electromagnetic Braking in Continuous Casting of Steel Slabs", *Metallurgical and Materials Transactions B*, 39B(1), pp. 94-107.
- [24] B. Rietow and B.G. Thomas, "Using Nail Board Experiments to Quantify Surface Velocity in the CC Mold", Proc. of AISTech 2008 Steelmaking Conf., Pittsburgh, PA, (2008).
- [25] H. Bai and B.G. Thomas, "Bubble Formation during Horizontal Gas Injection into Downward Flowing Liquid", *Metallurgical and Materials Trans. B*, 2001, vol. 32B(6), pp. 1143-1159.

 [26] G. Lee, B.G. Thomas and S. Kim, "Effect of Refractory Properties on Initial Bubble Formation in Continuous-casting Nozzles", *Metal and Materials International*, 2010, vol. 16(3), pp. 501-506.

CHAPTER 2.

MODEL OF GAS FLOW THROUGH POROUS REFRACTORY APPLIED TO AN UPPER TUNDISH NOZZLE

2.1. Introduction

Argon gas is widely used in metallurgical processes for many purposes, such as gas injection through a porous plug to stir the ladle, gas injection to remove inclusions in the tundish, and gas injection through the upper tundish nozzle (UTN) during continuous casting to prevent reoxidation and nozzle clogging^[1] as shown in Figure 2.1.^[2] This injected gas significantly affects flow in these vessels, and may be detrimental if not properly controlled. Extensive research has investigated gas-liquid two-phase interactions in those vessels (e.g. ladle, ^[3-6] tundish,^[7-8], and continuous casting^[9-14]), via physical and mathematical modeling. Physical models provide qualitative understanding of the gas-liquid two-phase interactions and can be used to validate computational models. However, physical model results from an air-water system differ from an otherwise similar argon-metal system due to some differences in material properties (e.g. surface tension and density) and in operation conditions (e.g. temperature gradient) between these two systems.^[10] Thus mathematical modeling becomes a necessary tool to study gas-metal two-phase flows in commercial processes. Computational models have been applied extensively to study argon gas effects on steel continuous casting using mixture models,^[10,14] Eulerian-Eulerian models^[9,11] and Eulerian-Lagrangian models.^[12-13] The accuracy of these modeling efforts depends on two key parameters: the volumetric flow rate of argon gas entering the steel in the hot condition, and the initial bubble size distribution. Both parameters have been investigated in previous work.^[10,14,16]

The volumetric flow rate of the injected argon gas is usually measured in the "cold", standard temperature and pressure (STP) condition well before entering the nozzle, in standard liters per minute (SLPM). This is usually much smaller than the flow rate entering the molten metal in hot condition through refractory walls, due to gas thermal expansion. This effect is accounted for with the ideal gas law, as implemented in [10] to estimate volumetric flow rate in the hot condition exiting the SEN port during continuous casting:

$$Q_{g,hot} = Q_{g,cold} \left(\frac{T_0}{T_\infty} \right) \left(\frac{p_\infty}{p_\infty + \rho g L_n} \right)$$
(2.1)

where Q_g is the gas flow rate (m³/s), T_0 is the casting temperature (K), T_{∞} is the ambient temperature (K), p_{∞} is the ambient pressure (Pa), and L_n is the pressure head of molten steel above the gas injection region (m). This calculated gas flow rate in the "hot" condition is then used to find the bubble size distribution and is applied as the inlet boundary condition for the gas phase in two-phase flow simulations.

This simple model relies on two basic assumptions: (1), no gas leakage, and (2), uniform pressure and temperature distributions at both the gas injection and exit surfaces. However, these two assumptions are not usually satisfied in real-world applications. Some of the gas escapes through leaks in the delivery system, cracks in the refractory or imperfect seals at the joints between refractory components. The remaining gas expands as it travels through the heated refractories and enters the steel flow with a non-uniform distribution. Operators can identify extreme leakage by the drop in the measured "back pressure" of the gas.^[15]

The bubble size distribution in the molten metal that results from gas injection is also important but difficult to determine. Iguchi et al.^[16-17] performed experiments with vertical gas injection into stagnant liquid in both mercury-air^[16] and iron-argon systems^[17] under relatively large gas flow rates (20-400 ml/s), and developed empirical correlations to predict gas bubble

sizes under different operating conditions. Bai and Thomas^[18] developed a semi-analytical twostage model to predict initial bubble formation from a single hole in the nozzle wall, considering the important drag effect of the downward-flowing steel. They calibrated and validated this model to reproduce measurements in air-water systems, and applied it to predict the bubble size entering into argon-steel flow systems. Ghaemi et al.^[19] recently measured the size distributions of microbubble formation upon entering a water channel with cross flow, taking into account the effects of the gas injector locations and also the bubble coalescence (termed secondary bubble formation) at high gas flow rates. Empirical correlations were generated to predict initial and secondary bubble sizes.^[19]

Little experimental work has been conducted to study gas flow through porous refractory and to investigate the bubble size distributions that exit the refractory surface to enter the liquid. Kazakis et al.^[20] injected air through metal spargers into initially stagnant water and measured the bubble size distributions. A correlation to predict the mean bubble size based on dimensionless groups including Froude (Fr), Weber (We), and Reynolds (Re) numbers was obtained from the air-water measurements.^[20]

In addition to the gas flow rate and pressure distributions, the prediction of bubble size distribution depends on quantifying the number density of "active sites"^[21] through which gas streams leave the porous refractory surface. Previous work^[22-23] has found that the number of active sites per unit area (#/cm²) is much fewer than the total available sites, and increases with increasing gas flow rate, and decreases with surface tension, and gas viscosity. In addition, the number of active sites depends on the specific permeability and porosity of the refractory,^[23] contact angle,^[23] cross-flow velocity,^[23] and liquid density.^[22]

Difficulties arise in correlating the gas flow rates and bubble size measurements in water models with those in metallurgical processes, due mainly to the great differences in temperature gradients and surface tension between the air-water and argon-steel systems.^[16,17] Also, to predict a reasonable initial bubble size distribution, the realistic, non-uniform distribution of the superficial (average) velocity of the gas exiting the refractory-liquid interface is needed, which cannot be obtained from experiments.

Thus a model is needed to predict realistic distributions of gas flow and velocity inside the porous refractory, taking into account the effects of non-uniform pressure and temperature, complicated geometry, refractory, gas, and interfacial properties, and joint sealing conditions, during bubble formation. In this work, new models and boundary conditions are developed to investigate these phenomena. After validation with both analytical solutions and experimental observations, the models are applied in a brief parametric study, and extensions to predict bubble size and gas leakage detection are discussed.

2.2. Mathematical Model Description

To model gas flow through a heated porous medium, the porous flow equations must be coupled with heat transfer and mass conservation to account for gas thermal expansion and temperature-dependent viscosity. To implement realistic boundary conditions, in this work, the pressure distribution of liquid metal surrounding porous refractory is calculated via Bernoulli's equation. In addition, a pressure threshold with a novel one-way flow condition is derived to account for interfacial tension effects during bubble formation at pore exits on the partially wetted surface of porous refractory. A general-purpose methodology to simulate gas flow through heated porous refractory is presented in the following sections.

2.2.1 Governing Equations

Two independent models are developed here to simulate gas flow through a porous medium: a pressure source model, and a complete porous-flow model. To incorporate the thermal expansion effect, a steady-state heat conduction equation was first solved to calculate the temperature distribution within the nozzle refractory walls:

$$\nabla \cdot \left(k \nabla T \right) = 0 \tag{2.2}$$

The gas density ρ is given by the ideal gas law:

$$\rho = p / (RT) \tag{2.3}$$

where *R* is the specific gas constant for argon, (207.85J kg⁻¹K⁻¹), *p* is absolute gas pressure (Pa), and *T* is absolute temperature (K).

To satisfy mass conservation, the following continuity Eq. (2.4) is solved in both models, which retains gas density because it is compressible and also varies due to thermal expansion,

$$\nabla \cdot (\rho \mathbf{v}) = 0 \tag{2.4}$$

where \mathbf{v} is the superficial (average) velocity vector of the gas flowing inside the porous refractory. The actual local velocity is much larger, depends on pore size, and is calculated from the results of this work only at the surface, knowing the number of active sites.

2.2.1.1 Porous-flow model

For the porous-flow model, the steady-state Navier-Stokes Eq. (2.5) are solved in three dimensions for the superficial gas velocities and pressure distribution with two additional momentum sink terms to simulate flow through porous media:

$$\rho \mathbf{v} \cdot (\nabla \mathbf{v}) = -\nabla p + \nabla \cdot (\mu \nabla \mathbf{v}) - \left(\frac{1}{K_D} \mathbf{v} + C \frac{1}{2} \rho |\mathbf{v}| \mathbf{v}\right)$$
(2.5)

The two extra sink terms in the brackets on the right-hand-side of Eq. (2.5) are first, for viscous momentum loss, and second, for inertial momentum loss, which is neglected in creeping flows (Re<1) by setting *C* to zero.^[24] When inertia is important, *C* could be measured from permeability tests^[25] or calculated from models.^[26]

The permeability of gas flow through a porous medium, K_D , is defined as the ratio of two properties, as shown in Eq. (2.6): the specific permeability K_{DS} of the refractory, and the dynamic viscosity μ of the gas, which varies greatly with temperature:

$$K_D = \frac{K_{DS}}{\mu(T)} \tag{2.6}$$

Here, K_{DS} is the refractory specific permeability, which is assumed to be isotropic in this work, and is given a typical value of 1.01×10^{-12} m². The specific permeability denotes the flow resistance of the porous refractory material, which depends on the pore structure connectivity of the porous medium, and does not depend on temperature, because solid thermal expansion effects on the pore structure are negligible.^[27]

2.2.1.2 Pressure-source model

The transition from diffusion to momentum-dominated flows in porous media (in both consolidated and unconsolidated cases) was found to start at a Re between 1.0~10, defined as:^[28-29]

$$\operatorname{Re} = \frac{Q\rho\delta}{\mu A\phi}$$
(2.7)

where Q is the fluid volumetric flow rate, ρ is fluid density, μ is dynamic viscosity, δ is the average pore diameter, A is the sample cross-section area, and ϕ is porosity. When the flow is laminar with low gas velocities, only the viscous resistance (Darcy's law) is needed to describe the flow in porous media. This is the situation in most cases of gas injection into liquid metal through porous refractories in metallurgical processes. Because the pores and flow rates are very small, flow is laminar over a wide range of ceramic refractory materials.^[27] For instance, for gas injection through UTN during continuous casting process, the typical Reynolds number of gas flow in the porous refractory calculated via Eq. (2.7) ^[28] is around 10⁻², which is much smaller than 1, so inertia is negligible. In this scenario of laminar gas flow, a simple "pressure-source" model is developed by adopting Darcy's law to obtain gas velocity distribution from local pressure gradient:

$$\mathbf{v} = -K_p \nabla p \tag{2.8}$$

Re-organizing Eq. (2.3), (2.4) and (2.8) gives Eq. (2.9), which is the final form of the equation solved in the "pressure-source" model.

$$\nabla \cdot \left(K_D \nabla p \right) = -\frac{RT}{p} \left[\nabla \left(\frac{p}{RT} \right) \cdot \left(K_D \nabla p \right) \right]$$
(2.9)

The left side of the Eq. (2.9) is a pressure diffusion term, and the right side contains two source terms to account for thermal expansion of the gas and the permeability (gas viscosity) change with temperature. This non-linear elliptic equation can be solved with a simple Poisson equation iterative solver, (same form as steady heat conduction with non-linear heat sources), so is easy to program with an in-house code. It is not noting that this pressure-source model loses accuracy when high superficial gas velocities are generated in the porous refractory (e.g.

Re>>1.0) and inertial effects become important. In such cases, the porous-flow model would be more accurate.

2.2.2 Boundary Conditions

The boundary conditions for the computational model include convective boundary conditions for the heat transfer analysis, pressure Dirichlet boundary conditions for the pressure injection slits (and exposed refractory area), zero normal-velocity conditions at sealed interfaces, and a novel pressure boundary condition to ensure one-way gas flow at the refractory-metal interface. Combining all of these boundary conditions enables the model to make realistic predictions for practical applications.

2.2.2.1 Heat transfer model boundary conditions

The boundary conditions on the heat transfer Eq. (2.2) depend on convection between the refractory and the surrounding flowing fluids. A convective boundary condition is applied at both the inner UTN surface contacting the liquid metal, and the outer surface surrounded by ambient air. The heat transfer coefficients, tabulated in Table II, are calculated using the following *Nu* number correlation from Sleicher and Rouse:^[30]

$$h = \frac{Nu \cdot k}{D}$$
, $Nu = 5 + 0.015 \,\mathrm{Re}^a \,\mathrm{Pr}^b$ (2.10)

where $a = 0.88 - \left(\frac{0.24}{4 + \Pr}\right)$, and $b = \frac{1}{3} + 0.5 \exp(-0.6\Pr)$, with $\Pr = \frac{\mu}{\rho \alpha}$, and $\operatorname{Re} = \frac{\rho UD}{\mu}$.

It is also important to note that the gas permeating through the micro-channels in the refractory quickly heats up to the local temperature of the UTN.^[9] So a "one-way" coupling of

the heat transfer model results to the gas flow model is adopted. This is another reasonable assumption of the current model.

2.2.2.2 Flow model boundary conditions

The boundary conditions for the porous gas flow simulations are an essential part of the model for realistic gas distribution and gas leakage predictions. On the surfaces of the verticalchannel and annular-shaped distribution slits in the refractory, the gas injection pressure is fixed:

$$p = p_{inject} \tag{2.11}$$

The interfaces between the refractory and the outer steel can are assumed to be perfectly sealed to prevent any leakage, so also have zero normal gas flow, which is imposed in the porous-flow model as follows.

$$\mathbf{v} \cdot \mathbf{n} = 0 \tag{2.12}$$

where \mathbf{n} is the outward-normal direction vector at the appropriate domain boundary. The following equivalent condition is imposed to prevent normal flow for the pressure-source model:

$$-K_{D}\left(\nabla p\right)_{b}\cdot\mathbf{n} = -K_{D}\left(\frac{\partial p}{\partial n}\right)_{b} = 0$$
(2.13)

On symmetry planes, this "no penetration" condition must again be applied, which is equivalent to a Neumann B.C.^[31] for pressure, with zero surface-normal pressure gradient, as given in Eq. (2.12) or (2.13).

For refractory surfaces that are exposed to the surrounding environment, the ambient pressure is used as the boundary condition, given by:

$$p = p_{ambient} \tag{2.14}$$

At joints between refractory pieces, both the sealed-bottom and open-bottom cases are explored, and boundary conditions in Eq. (2.13) and (2.14) are adopted respectively for these two cases.

2.2.2.3 Refractory-metal interface

The interface between the porous refractory and the liquid (metal) is where the most important and influential boundary condition of the model must be applied. For gas to exit the refractory-liquid interface into the liquid bulk, a pressure jump is needed to overcome the surface tension force in order to curve the interface, form and detach bubbles. This pressure jump Δp is a "bubbling" threshold that can be calculated as:

$$\Delta p = p_b - p_l = \sigma \kappa \tag{2.15}$$

where p_b is gas pressure p at the refractory interface (or boundary), p_l is liquid pressure at the interface, σ is the surface tension, and κ is the curvature of the bubble surface at the refractory-liquid interface. This quantifies how the liquid pressure is different from the pressure of the gas inside the refractory.

Figure 2.2 illustrates the stages of bubble formation. At the initial stage 1, less than a half of a spherical bubble surface intrudes into the liquid and the radius of the bubble cap is larger than that of the pore opening, resulting in a small curvature. At stage 2, the hemisphere bubble diameter equals the pore diameter which defines the maximum curvature. At the third stage, the bubble has expanded beyond the hemisphere shape, with a larger diameter and less curvature.

The pressure jump is governed by the maximum bubble surface curvature, which occurs at the second stage, and can be rewritten in terms of the refractory pore size as follows:

$$\kappa = \frac{2}{r_{pore}} \tag{2.16}$$

where r_{pore} is the radius of the pore opening on the UTN inner surface (assumed to be circular). The pressure jump threshold also governs flow entry into the refractory pores from the liquid metal. Kaptay et al.^[32] studied this "liquid penetration" phenomenon experimentally and tabulated the pressure threshold for liquid to enter the porous refractory and the maximum penetration depth. A capillary pressure balance needs to be considered to predict this maximum liquid penetration depth, which includes the triple point contact angle between the liquid, gas, and refractory inside the pore, in addition to the pore diameter. During continuous casting, this threshold pressure for liquid steel penetration is around 42 kPa, for a pore diameter of 100 µm and a contact angle of 150° .^[33] Considering the low liquid pressures that accompany gravity-driven flow, this result implies that the liquid steel should never enter the UTN refractory in metallurgical processes with typical gas injection pressures.

Based on the discussions above, the boundary condition at the refractory-liquid interface must serve two different scenarios: when gas pressure near the interface exceeds the pressure threshold for bubble formation, the boundary pressure should equal the sum of the local liquid-steel pressure and the bubbling pressure threshold; when the gas pressure is smaller than the bubbling pressure barrier, the boundary should be considered "sealed" as no liquid steel penetrates into the refractory and no gas exits the local refractory pores. The latter zero-penetration velocity boundary condition can also be satisfied by applying Eq. (2.13).

In this work, a novel Robin-type^[31] (or mixed) boundary condition (see Appendix D), is developed to satisfy the two requirements of the previous discussion, by enforcing the following equation set for one-way flow at the refractory-liquid interface:

$$\begin{cases} if \quad \left(\frac{\partial p}{\partial n}\right)_{b} < 0, \qquad p_{b} = p_{l} + \sigma \kappa \\ else, \qquad \qquad \left(\frac{\partial p}{\partial n}\right)_{b} = 0 \end{cases}$$
(2.17)

where p varies with distance along the interface boundary (subscript *b*). This equation seals portions of the boundary, which are unknown prior to the calculation. The choice of Dirichlet (first) or Neumann (second)^[31] boundary condition case in Eq. (2.17) is part of the model solution. Clearly, iterative methods are needed to fulfill this purpose, as discussed later.

2.2.3 Liquid Pressure Model

The boundary condition, Equation 17 requires p_l , which depends on the behavior of the liquid outside of the refractory domain. However, the pressure distribution in the liquid metal usually cannot be measured directly in the metallurgical vessels, and may be difficult to obtain. Sometimes, such as in a ladle porous plug, this pressure can be considered a constant as the hydrostatic pressure. In other situations, such as beneath a stopper rod, a full computational model of the turbulent fluid flow of the liquid steel should be applied. In this work, a simple model to find p_l with vertical distance down the UTN wall was found using Bernoulli's equation, based on balancing potential and kinetic energy of the fluid flow:

$$p(z) = p_0 + \rho_l g(h_{tundish} + h_{UTN} - z) - \frac{1}{2}\rho_l U^2$$
(2.18)

where p_0 is the pressure at tundish level (atmospheric pressure), g is gravitational acceleration, $h_{tundish}$ and h_{UTN} are heights of the tundish level and UTN respectively, U is mean velocity at any UTN cross-section, and z is the distance above the UTN bottom. This equation is reasonable for any distance above the slide plate, below which the pressure drops due to the sudden contraction of the flow area) and U increases in a non-uniform manner.

2.2.4 Material Properties

The permeability depends on both the pore structures of the consolidated refractory, and the dynamic viscosity of the gas, μ , which varies greatly with temperature. Thus, in metallurgical processes involving large temperature gradients, the gas viscosity and permeability are coupled with the temperature field. Incorporating anisotropic, temperature-dependent specific permeability and non-uniform pore size distribution into the current model is straight forward. Lacking such property measurements, this work assumes isotropic, temperature-independent specific permeability and uniform pore size. The argon gas viscosity is taken from measurements [34].

$$\mu(T) = \mu_0 * 10^{(0.63842\log_{10} T - 6.9365/T - 3374.72/T^2 - 1.51196)}$$
(2.19)

where μ_0 , 2.228×10⁻⁵ Pa·s, is the dynamic viscosity at 293 K (20 °C).

Thermal conductivity of the refractory is 33 W/(mK). The surface tension between the liquid steel and argon gas is 1.157 N/m. A typical specific permeability of 10.1 npm is chosen for the base case, taken from measurements on a medium-permeability refractory with 17% porosity.^[23,35] For a parametric study, this specific permeability is varied between 2.0 and 12.0 npm.

2.2.5 Numerical Details

Owing to the one-way coupling of this system, the energy Eq. (2.2) is first discretized using a finite volume method and solved for the temperature field, followed by solving for the pressure and velocity fields. For the porous-flow model, the coupled ideal gas law and momentum Eq. (2.3)-(2.5) were discretized using the third-order MUSCL scheme^[36] and solved

using the SIMPLE algorithm^[37] in Fluent by ANSYS Inc.^[24] For the pressure-source model, Eq. (2.9) was similarly discretized and solved using the User Defined Scalar (UDS) function in Fluent,^[24] with the non-linear source term right side of Eq. (2.9) implemented in a User Defined Function (UDF). Velocities are then computed from the pressure field using Eq. (2.8). The system converges easily with any solution strategy.

The one-way flow pressure boundary condition is applied in an iterative process. At the beginning of each new iteration i+1, a pressure is prescribed at the local boundary face according to the evaluation of conditions in Equations 17, based on the known pressure distribution from previous iteration, *i*, via Eq. (2.20) below:

$$\begin{cases} if \quad \left(\frac{\partial p}{\partial n}\right)_{b}^{i} < 0, \qquad p_{b}^{i+1} = p_{l} + \sigma \kappa \\ if \quad \left(\frac{\partial p}{\partial n}\right)_{b}^{i} \ge 0, \qquad \left(\frac{\partial p}{\partial n}\right)_{b}^{i+1} = 0 \end{cases}$$

$$(2.20)$$

The Dirichlet case in Eq. (2.20) is straight forward to apply. The zero pressure gradient condition normal to the surface is achieved by setting the pressure on the local surface, according to Eq. (2.21), which is derived from the zero diffusion flux boundary condition for an unstructured mesh in finite-volume formulation from Mathur and Muthy ^[38] and adopted in Ansys Fluent,^[24] to account for non-orthogonal boundary cells.

$$p_{b} = p_{c} + d_{b} \left[\left(\nabla p \right)_{c} \cdot \mathbf{e}_{b} - \left(\nabla p \right)_{c} \cdot \mathbf{A}_{b} \frac{\mathbf{A}_{b} \cdot \mathbf{e}_{b}}{\mathbf{A}_{b} \cdot \mathbf{A}_{b}} \right]$$
(2.21)

where subscript *c* represents the cell adjacent to the current boundary face (with subscript *b*); \mathbf{e}_b is a vector from the cell centroid to the geometric center of the boundary face with magnitude equal to the distance, d_b ; "area vector" **A** is a vector perpendicular to the cell boundary face with magnitude equal to the face area. If vectors \mathbf{e}_b and **A** are parallel (indicating that the cell is

orthogonal), then Eq. (2.20) simplifies to $p_b = p_c$. Further computational details are presented elsewhere ^[38-39] and in later sections.

2.3 Model Validation

Two separate test problems related to steel processing were used to validate the models described in the previous section: 1), a simple one-dimensional problem with analytical solution, and 2), a real bubbling experiment in a submerged commercial UTN.

2.3.1. Comparison with 1-D Analytical Solutions

The first test problem is one-dimensional cylindrical flow of argon gas, which is injected into the outer-radius surface of a round refractory pipe, and exits from the inner-radius surface. Figure 2.3 shows the computational domain of one quarter of a short segment of the pipe, and boundary conditions investigated for this test problem. The mesh contains 60,000 hexahedral cells in a Cartesian coordinate system. The problem is solved numerically in three dimensions, with both the pressure-source model and the porous-flow model.

The governing Eq. (2.2) and (2.9) simplify to the following coupled ordinary differential equations (ODEs) in a cylindrical coordinate system with respect to radial position, *r*,

$$\begin{cases} \frac{1}{r}\frac{d}{dr}\left(r\frac{dT}{dr}\right) = 0\\ \frac{d^2p}{dr^2} + \frac{1}{r}\frac{dp}{dr} = \left(\frac{1}{T}\frac{dT}{dr} - \frac{1}{K_D}\frac{dK_D}{dr}\right)\frac{dp}{dr} - \frac{1}{p}\left(\frac{dp}{dr}\right)^2 \end{cases}$$
(2.22)

The outer surface boundary condition is either specified pressure p (case 1) or specified velocity V (case 2). The inner-radius surface has constant absolute pressure of 100kPa as boundary condition in both cases. The inner and outer surface temperatures are fixed at T_1 and T_2

respectively. The two side walls, and the top and bottom walls of the numerical model are set to symmetry planes. The parameters, tabulated in Table I, are chosen to approximate realistic operating conditions of a typical UTN during continuous casting.

In both case 1 and case 2, three different scenarios are considered to study the effects of temperature-dependent gas viscosity and gas compressibility: (1) without any thermal effects; (2) with thermal expansion and constant gas viscosity; and 3) with thermal expansion and a realistic temperature-dependent gas viscosity. The numerical simulation results are compared with the analytical solutions derived in Appendix C.

The temperature solutions are compared in Figure 2.4, which also shows the realistic gas viscosity profile. The radial heat flow causes a slight deviation from a linear temperature profile, and the numerical simulations of this simple problem is exact to within 0.02% (based on L2-norm).

Pressure results from using the fixed pressure boundary condition are shown in Figure 2.5, for the three different scenarios, and compared with their respective analytical solutions. Three curves each are presented for the porous flow model, (squares), the pressure-source model (circles), and the 1-D analytical solutions, (lines). For all three cases, the pressure-source model and porous-flow model match perfectly with each other, and with the analytical solutions. Thus all three models are validated. Thermal expansion and temperature-dependent gas viscosity both produce higher pressures everywhere within the nozzle. Without thermal effects, the pressure distribution is almost flat, with slight upward curvature caused by the surface area difference between the inner and outer surfaces of the curved refractory walls.

Pressure distributions with the fixed argon mass-flow-rate (velocity) boundary condition are shown in Figure 2.6. Again, the two numerical models and the analytical solution all agree almost exactly for each of the five scenarios tested, providing further model validation. To achieve the same gas flow rate at the outer surface, the pressure inside the wall must increase to overcome three different effects which combine together, in order of importance: thermal expansion of the gas, increasing gas viscosity with temperature, and difference between inner/outer tube diameter. Increasing temperature towards the inside-radius tube bore causes the gas to expand, and the gas viscosity to increase. Results for constant gas viscosity lowers the different temperatures (293 K, 1000 K and 1800 K) show that increasing gas viscosity lowers the permeability, which requires a higher injection pressure to enforce the same gas flow rate. Finally, the smaller area of the inner surface requires a higher pressure to push the same amount of gas through the inner surface of the refractory into the liquid, relative to a flat wall.

2.3.1 Comparison with UTN bubbling experiment

The second test problem is a bubbling experiment, which was carried out in a commercial UTN submerged in water.^[40] Porous-flow model results are compared with observations of bubbles exiting the inner surface. The UTN was cut in half, sealed at the cut surface, and placed in a water tank. Gas was injected into the bottom of the vertical slit, indicated with dashed lines in Figure 2.7(a). Note that the UTN in this test was cut in half perpendicular to the cut for the symmetrical half-nozzle domain used in the simulation, as shown in Figure 2.7(b).

During the experiment, injection pressure was increased slowly, so the gas flow rate increased gradually from zero. The UTN was tilted so that the rising bubbles would not block observation of the "active sites" on the refractory inner surface where bubbles emerge. Because of the good wettability between UTN refractory and water, the hydrostatic pressure along the nozzle, and capillary effects, water may enter some of the pores at the UTN inner surface and block the pore exits. Thus, the bubbling threshold in this test prevents bubbles from exiting regions of low gas pressure.

Both the experiment and the model show a very non-uniform distribution of air exiting the UTN inner surface, as shown in Figure 2.7. The velocity contours in Figure 2.8(a) show that gas velocity is greatest at the injection slits, decreases as it diffuses through the refractory, and then increases again when approaching the UTN inner surface. The velocity vectors in Figure 2.8(b) show where the surface velocity drops to zero on certain regions of the UTN inner surface far from the injection slits near the bottom. This is due to the pressure threshold enforced by the one-way flow boundary condition. The local pressure in this part of the nozzle was never large enough to overcome the threshold.

It is important to note that the regions where no bubbles are observed in Figure 2.7(a) match well with the zero-velocity regions in Figure 2.7(b). There is a slight mismatch near the UTN bottom region where the photo shows gas bubbles coming out from UTN inner surface, while simulation shows no bubbles in that region. One possible explanation for this discrepancy is a nonuniform pore size produced in the UTN refractory during its manufacture: larger pores near the nozzle bottom surface would lower the pressure "bubbling" threshold. The high-velocity regions in Figure 2.7(b) match with regions where large bubbles are observed in Figure 2.7(a). Overall, a reasonable match was found between the simulation and the experimental observations, which validates the model, including the one-way-flow pressure boundary condition.

2.4. Model Application to UTN

34

The validated porous-flow model is next applied to study gas flow distribution in a typical double-slitted UTN shown in Figure 2.9, under realistic plant operating conditions. A gas line delivers argon into the UTN via a system of internal grooves (slits) that distribute the gas into the porous refractory, as pictured in Figure 2.1. Argon gas is injected into this UTN near the bottom of a vertical slit which feeds two ring-shaped (annular) slits in the upper and lower parts of the nozzle. The vertical slit is needed to reach the upper annular slit, which is inside the tundish bottom and not easily accessible to a gas injection line. During operation, the large temperature gradients in the flowing steel system cause thermal strains that may separate the joints between the refractory parts, such as the UTN and the upper plate. Opening a gap at a joint leads to a local pressure drop and possible gas leakage. The porous-flow model was then applied to investigate gas leakage by comparing two extreme conditions at the UTN bottom surface: one perfectly sealed, with the zero penetrating flow condition, (Eq. 2.13) (base case), and the other with a large gap, or completely open bottom, with the constant pressure condition (Eq. 2.14). Note that in order to be consistent with plant pressure measurements, all the pressure values in this section refer to the gauge pressure relative to the ambient pressure of 1 atm (101kPa).

The computational domain is a radial slice through this UTN, assuming axisymmetry, and the two-dimensional mesh (cylindrical coordinates) contains 5000 4-node quadrilateral cells. They are displayed in Figure 2.10, together with the boundary conditions. The base case, used in these parametric studies adopts the temperature-dependent gas viscosity in Eq. (2.19). The liquid steel pressure distribution at the refractory-liquid interface determines the overall resistance to gas flow for a given injection pressure. For a slide-gate flow-control system, the UTN is between top of the upper plate (Figure 2.1) and tundish bottom. Liquid steel flows from the tundish through the UTN, the holes in the three plates, and the submerged entry nozzle (SEN) into the

mold. The static pressure distribution from the liquid steel along the inside surface of the UTN, p_b is reasonably calculated as the hydrostatic pressure with Eq. (2.18), assuming uniform average vertical cross-flow velocity from the flow rate, and is chosen for the base case, as plotted in Figure 2.10. The distribution is nearly linear with a maximum pressure of 56 kPa at the UTN bottom just above the upper plate. This base case also adopts a sealed bottom (no leakage at joint), the bubbling pressure threshold, and the one-way flow boundary condition at refractory-liquid interface via Eq. (2.17), in which κ is calculated from Eq. (2.15). Other operating conditions for the base case are given in Tables II and III.

The temperature field from the heat transfer model is shown in Figure 2.11. Naturally, temperature increases almost linearly towards the UTN center. The 3-fold temperature increase causes significant gas expansion, which greatly affects the flow results.

Different boundary conditions on the UTN inner surface were investigated to further demonstrate and evaluate the computational model. Then, parametric studies were conducted to investigate the effects of possible joint (between the UTN and the upper plate) gas leakage, injection pressure and refractory permeability on the pressure and gas velocity distributions. The gas leakage fraction under different injection pressures was extracted and used to evaluate the performance of this UTN design.

2.4.1. Effect of Pressure Condition at Refractory-Liquid Interface

Three different pressure distributions in the liquid steel along the refractory-liquid interface are compared to investigate the importance of this boundary condition. Case 1 is the realistic base case, which has a sealed nozzle bottom (no gas leakage at the UTN-plate joint), the linear p_l in Eq. (2.18), and the gas-bubble pressure threshold, in Eq. (2.16). Case 2 is identical to

Case 1, but without the bubbling pressure threshold (setting κ to 0 in Eq. 2.15). Case 3 has an open bottom, and no pressure threshold for bubble formation at refractory-liquid interface. This case simply assumes constant ambient pressure of 1 atm, (0 Pa gauge pressure) for both the interface, p_l , and UTN bottom.

The gas pressure distributions calculated for these three cases are shown in Figure 2.12. Case 1 (the base case) in Figure 2.12(a) shows the smallest pressure gradients across the refractory of the three cases. This is because the bubbling threshold and higher liquid hydrostatic pressure combine to increase the resistance to gas flow, which reduces the pressure drop. Case 2 in Figure 2.12(b) has larger pressure gradients, especially near the upper slit, due to the lower liquid hydrostatic pressure towards the nozzle top. Case 3 in Figure 2.12(c) has the steepest pressure gradients.

Velocity distributions exiting the UTN are plotted in Figure 2.13 for the three cases. A peak in the velocity profile occurs near each of the two slits, simply due to the close proximity to the gas injection slit, which increases the pressure gradient. The realistic base case 1 has the lowest exiting velocities of the three cases, due to the bubbling threshold. Without this threshold, Case 2 has gas exiting velocities increased by ~5 times. Case 3 has the highest exit velocity because the giant pressure gradient from the lower injection slit to the open bottom increases the gas flow everywhere. An additional case, Case 4, is included to show the effect of temperature-dependent gas viscosity in the open bottom case (Case 3), which is small if the temperature for the constant viscosity is chosen to be the liquid metal temperature.

2.4.2. Effect of One-Way Flow Pressure Boundary Condition

The velocity field, with and without the proper one-way flow condition is compared in Figure 2.14 near the UTN bottom, which is open and leaks gas into the joint. Conditions are identical to those of the base case, except for the open bottom, and lack of the one-way flow condition for the case in Figure 2.14(b). With the one-way flow condition, gas exits the refractory only through the UTN bottom, as shown in Figure 2.14(a). Without this one-way condition, reverse flow occurs, as shown in Figure 2.14(b). This unphysical result is caused by the liquid pressure near UTN bottom exceeding the local gas pressure. As discussed before that the location on the UTN inner surface where the pressure boundary becomes a "sealed" wall is embedded as part of the solution from the model. The comparison in Figure 2.14 indicates that this model is able to seal the regions where liquid pressure is larger than the resolved local gas pressure, and thus prevent the large unphysical "reversed flow" from occurring. These results show that the one-way flow pressure boundary condition is important to realistic model predictions.

2.4.3. Effects of Joint Sealing Conditions

The effect of joint leakage is investigated first by comparing the pressure and velocity distributions of an open bottom case (Figure 2.15a) with the base case (Figure 2.15b), for otherwise identical conditions. Similar pressure and gas velocity distributions are found near the upper slit of the UTN in both cases. However, the distributions near the UTN bottom are very different, as expected. With an open bottom, pressure drops from ~100 kPa to 0 Pa (guage pressure) within a very short distance (from the lower slit to the open bottom), which generates a huge gas velocity exiting the domain. With the one-way flow pressure B.C., the UTN inner surface near the bottom is sealed, preventing any liquid/gas penetration into the porous refractory.

Although the injection pressure near the lower slit is higher than the local liquid pressure at the refractory-liquid steel interface, no gas exits that interface near the bottom of the UTN, since the open bottom draws in all the flow and causes the pressure there to drop below the local liquid pressure near the UTN bottom. Gas exits the path of least resistance as shown in the zoomed-in velocity plot in Figure 2.15(a). With perfect bottom sealing, Figure 2.15(b) shows how gas can only exit through the UTN inner surface into the liquid steel stream. The corresponding pressure field near the sealed bottom then becomes almost constant, resulting in very low gas velocities. Note that the reference vector scale in Figure 2.15(b) is increased 10 times for better visualization.

The velocity profiles exiting along the UTN inner surface are plotted for both cases in Figure 2.16. Flow near the upper slit is almost identical for both cases. However, the flow drops to zero about 80 mm from the UTN bottom (about 1/3 of the total UTN length) in the open bottom case. This is because of the huge pressure gradient created by the imposed ambient pressure at the open bottom, which lowers the flow resistance and deflects the gas flow towards the nozzle bottom.

To quantify and evaluate the performance of the UTN design, the leakage fraction, θ_L , is the amount of the gas leaking away relative to the total gas injected.

$$\theta_L = 1 - \frac{m_{in}}{\dot{m}_{total}} \tag{2.23}$$

where \dot{m}_{in} is the gas mass flow rate entering the liquid steel found by integrating the velocity distributions over the UTN inner surface area, and \dot{m}_{total} is the total mass flow rate of the injected gas integrated over the injection-slit surfaces. For the open bottom case in Figure 2.16, the leakage fraction is 86%. This is because a very high flow rate through the leaking open bottom is needed to maintain the pressure gradient. This gas is not wasted because it helps, together with

gas injection into the plate, to flush the joint to lessen air aspiration. However, the leakage should be lessened by tighter sealing of the joint, using non-porous refractory, moving the lower gas distribution slit further above the nozzle bottom to lessen the pressure gradient, or other means.

Another type of leakage, possibly caused by thermal expansion differences, is separation of the porous refractory from the outer steel container or "can" that seals the gas distribution slits. This may allow gas from the distribution slits to spread everywhere around the thin gap that forms between the outer surface of the refractory and the can. A simulation of this situation (not shown) resulted in a higher total gas flow rate with a more uniform distribution of gas exiting from the UTN inner surface than for the other cases in Figure 2.13. This situation also increases the danger of extra gas leakage from any openings of the thin gap to the ambient atmosphere. It also increases the rate of leakage through the joint at the nozzle bottom, if it is not perfectly sealed.

2.4.4. Effect of Injection Pressure

Injection pressure drives gas to diffuse through the UTN refractory and is routinely measured during plant operations. A parametric study varying injection pressure was conducted for both the perfectly sealed base case and the same case with an open bottom, with other conditions given in Table III.

Pressure contours for cases with three different injection pressures (90, 99 and 140 kPa) are compared in Figure 2.17 for the perfectly-sealed base case. If the injection pressure is less than a critical minimum injection pressure, then the entire UTN inner surface acts like a "sealed" wall with no gas flowing out, due to the one-way flow boundary condition and pressure threshold

at this interface. The pressure and gas velocity distributions for cases chosen near this critical injection pressure are shown in Figure 2.17(a). Most of the refractory is at an almost uniform pressure near the gas injection pressure of 90 kPa, with almost no pressure gradients. Gas exits only from a tiny region near the very top of the UTN, where the liquid pressure is lowest, and at a negligible flow rate. Increasing the injection pressure to 99 kPa enables gas to exit from the upper half of the UTN inner surface, as shown in Figure 2.17(b). No gas exits from the lower half because the bubbling threshold is too high, due to the increased liquid hydrostatic pressure towards the nozzle bottom. Further increasing the injection pressure to 140 kPa, gas exits from everywhere on the UTN inner surface, as shown in Figure 2.17(c).

Figure 2.18 shows the velocity distributions leaving the UTN inner surface along the axial direction for four different injection pressures (ranging from 90 kPa to 140 kPa) and two different UTN joint sealing conditions (open bottom and perfectly sealed). For a given injection pressure, the velocity profiles for the open bottom and perfectly sealed cases are similar in the upper half of the UTN. Near the UTN bottom, however, the flow drops to 0 for all of the open-bottom cases. This is because the low ambient pressure (1 atm) at UTN bottom is less than the gas pressure threshold at the UTN inner surface. With perfect sealing, for higher injection pressures (120 and 140 kPa gauge, e.g.), a second velocity peak is found near the UTN bottom. This is due to high gas velocity exiting regions close to the lower injection slit, as previously discussed.

Figure 2.19 shows the change of calculated gas mass flow rate at injection slits and that enters liquid steel (flowing through UTN inner surface) with different injection pressures, for both open bottom and perfectly sealed cases, corresponding to the velocity profiles shown in Figure 2.18. Total injected gas flow rates increase with the injection pressure and are higher in the open bottom case (circles) compared to the perfectly sealed case (squares) for each of the injection pressures studied. The total injected gas flow rate in the perfectly sealed case (squares) increases non-linearly with the injection pressure: for a 9 kPa increase of the injection pressure from 90 kPa to 99 kPa gauge close to the gas exiting threshold, injected gas flow rate increases by 2 orders of magnitude, from $\sim 3 \times 10^{-7}$ kg/s to $\sim 3 \times 10^{-5}$ kg/s. As shown also in Figure 2.19, gas mass flow rate entering liquid steel in the open bottom case increases with the injection pressure, with a trend similar to that in the perfectly sealed case. The ratio of gas flow rate entering liquid steel in the open bottom case (triangles) to that in the perfectly sealed case (squares) stays ~0.7 consistently under all injection pressures higher than 99 kPa (above the gas exiting injection pressure threshold). Results with calculated gas leakage fraction following Eq. (2.23) show that the amount of gas leaking into joint increases from 70% with an injection pressure of 140 kPa to ~97% with a 99 kPa injection pressure. With a 90 kPa injection pressure in the open bottom case, all of the gas injected into the UTN ($\sim 5 \times 10^{-4}$ kg/s) leaks into the joint.

Figure 2.19 shows that the gas flow rate lost from the open bottom due to leakage is always very large, exceeding 70%. This finding may be specific for the current UTN design, however, which has an annular slit located close to the leaking joint. However, the total gas flow rate increases so much for the leaking joint cases, that the gas flow rate exiting into the liquid steel always exceeds 70% of the sealed-bottom case, assuming that the pressure can be maintained (at 99kPa gauge or above).

2.4.5. Effect of Permeability

Specific permeability of the porous refractory indicates its resistance to fluid flow through the spaces in its pore structure. The effect of specific permeability on gas velocity distribution is investigated for both the perfectly-sealed bottom base case, and the same case with an open-bottom (leaking joint). Figure 2.20 shows profiles of the gas velocity exiting along the UTN inner surface for both cases. Figure 2.21 summarizes the effects of specific permeability on the total gas flow rate exiting the UTN into the liquid steel, based on integrating the velocity profiles in Figure 2.20. Increasing the specific permeability causes linear increase of the gas exiting velocities from the UTN inner surface (or at the joint in the open-bottom case). The shape of the velocity profile does not change, however, owing to the simple linear relationship between pressure gradient and flow given in Darcy's law Eq. (2.8) and because varying specific permeability does not change the pressure distribution. This is expected because K_{DS} can be cancelled from K_D on both sides of Eq. (2.9), if it has no spatial variation. Thus, the total gas flow rate and the flow rate entering liquid steel both increase linearly with increasing specific permeability for both perfectly-sealed and open-bottom cases, as shown in Figure 2.21.

With an open-bottom (dashed lines), gas exit velocities all drop to 0 near the bottom of the UTN inner surface. It is easier for the gas to escape from the bottom surface, than to overcome the threshold pressure jump needed to form a bubble at the interface. Figure 2.21 shows that 86% of the gas leaks from the open bottom in Case 2. This large amount does not depend on permeability, but is expected to depend on geometry of the refractory and its slits. Raising the lower distribution slit further away from the UTN bottom joint, or better sealing would lower this leakage fraction.

2.4.6. Practical Applications

A comprehensive modeling system is proposed to estimate the argon gas flow distribution and initial bubble size entering the liquid steel through nozzles with porous

43

refractories. The porous gas flow model introduced in this work is the first step of this system. This model calculates superficial gas velocity distributions on the UTN inner surface. Next, an empirical model from previous water model bubbling experiments^[23] is used to estimate the number density of active sites, based on the local average gas velocity from the first step. Then, the hot gas flow rate from each active site is calculated from the local gas velocity and the local density of active sites. The calculated macroscopic superficial gas velocity is then converted to the physical gas velocity at pore-openings. The gas flow rate through each pore (active site) is input to a two-stage model of bubble formation in downward flowing liquid^[18] to calculate the average bubble size entering into the liquid steel. Finally, a multiphase model of turbulent fluid flow in the nozzle and mold can use these results in realistic simulations to solve practical problems.

2.4.6.1.Gas leakage detection

When argon gas is injected during metallurgical processes such as continuous casting, both the volumetric flow rate of gas (usually in SLPM) and the injection pressure are measured and recorded. However, the argon mass flow rate entering the liquid steel can be determined using either one of these two measurements, provided that the specific permeability of the refractory is known. In this work, the measured injection pressure is the preferred boundary condition, because it indicates the true resistance needed to push the heated gas into the liquid, and drops if there is leakage. The volumetric flow rate, on the other hand, is usually measured upstream before any gas leakage has occurred, so often overestimates the total gas flow entering the molten steel. Combining the two measurements together enables extra information to be gained about the process. If both the (cold) flow rate and the back pressure are correctly measured, then the actual gas flow rate calculated with the model can be compared with the measured flow rate to quantify how much gas has leaked. An example using this model system to detect gas leakage during continuous casting process can be found elsewhere.^[39]

2.4.6.2.Design of slide-gate and stopper-rod gas delivery systems

The liquid pressure distribution over the UTN inner surface in a stopper-rod system is very different from that in the slide-gate system studied in this work. In a stopper-rod system, the gas usually is injected through the stopper-rod tip or the porous refractory of the upper UTN into a very low-pressure region below the gap between the stopper-rod tip and the UTN wall. This causes a high gas-velocity region with high bubble concentrations or even gas pockets, leading to complicated two-phase interactions such as annular flow and other problems.^[40] If the upper annular slit is too far above the attachment point of the gas injection line, the needed long vertical slits may lead to asymmetric flow.^[40] Careful model calculations of the pressure distribution are needed to predict gas flow behavior and bubble size in this situation.

For slide-gate systems, the results of this work suggest that, the annular gas distribution slit should be located far away from joints at the UTN bottom, to avoid or reduce possible gas leakage from the joint surface. In addition, the optimal location of the gas distribution slits, and the choice of porous / non-porous refractory could be obtained via studies using the porous-flow model developed in this work.

2.5. Conclusions

A general-purpose model of gas flow through porous refractory has been developed in this work, which includes the pressure threshold at the refractory / liquid metal interface due to gas bubble formation, and a new one-way flow pressure boundary condition to ensure no improper liquid penetration into the refractory. The following conclusions are drawn from initial parametric studies with this new approach:

- Model validation with 1-D analytical solutions shows that both versions of the model correctly account for the effects of thermal expansion of the gas, temperature-dependent gas viscosity, and geometric effects in porous refractory flow problems with high temperature gradients.
- 2. Model validation with water bubbling experiments shows that the model can accurately simulate gas flow distributions exiting real nozzle refractories, which involve relatively slow gas flow rates, in the regime where Darcy's law holds.
- 3. A critical injection pressure exists to allow gas bubble formation on the UTN inner surface. Bubbles form and exit from only a fraction of the surface area, where the pressure gradients are highest. This surface area fraction naturally increases with gas injection pressure.
- 4. The gas velocity profile varies greatly over the UTN inner surface, according to the nozzle geometry and the location of the gas distribution slits. More bubbles exit near to the annular slits, and where the external liquid pressure is lower, owing to the higher pressure gradients.
- 5. A large amount of gas may escape from the UTN bottom, if the joint leaks. This leakage fraction is predicted to exceed 80% for the particular design studied, which has an

annular gas distribution slit relatively near to the bottom joint, and no non-porous refractory.

- 6. Increasing the gas injection pressure naturally increases the total gas flow rate injected. It also decreases the leakage fraction, owing to more gas flowing from the upper slit, for this particular design.
- 7. The pressure distribution in the refractory is not affected by the specific permeability of the refractory, if it does not vary spatially. Therefore, the total injected gas flow rate increases linearly with increasing specific permeability, while both the shape of the velocity profiles and the leakage fraction stay constant.
- 8. The seal between the gas distribution slits and the outer-containment steel can is important to the flow distribution. Faulty sealing would lead to increased, more uniform flow distributions but more susceptibility to gas leakage problems.

The new model presented here can serve as the first step of a comprehensive modeling system to simulate gas-metal two-phase flow more accurately. It can provide realistic gas flow distributions in real metallurgical systems, for better estimation of the nonuniform bubble size distributions. Given the measured injection pressure and gas flow rate, this model can also predict the fraction of gas leakage in the real process.
2.6. Tables and Figures

Table I. Parameters for 1-D Test Case Boundary Conditions (based on absolute pressure)

R ₁ (m)	R ₂ (m)	P_{1} (kPa)	P_{2} (kPa)	T ₁ (K)	T ₂ (K)	V (m/s)
0.0375	0.0725	100	200	1800	1000	0.0073

Table II. Parameters for UTN Heat Transfer Analysis

Heat transfer	coefficients	Thermal	Viscosity	Density	Mean steel
h_{inner}	h_{outer} (W/m ² K)	conductivity $K_{\rm c}$ (W/mK)	μ	ρ (leg/m ³)	velocity, U
(W/III K)	(••/ •• • • •	\mathbf{K}_{s} (W/IIIK)	(Pa ⁺ s)	(kg/m)	(11/8)
25000	10	33	0.0056	7200	1.6

Table III. Refractory Properties and Casting Conditions in Parametric Study (Base case)

Tundish level (m)	Surface Tension σ (N/m)	Injection Pressure (Gauge) (kPa)	Ambient Pressure	Mean Pore Radius (µm)	Permeability (npm)
			(Gauge) (kPa)		
0.70	1.157	110	0	50	10.1



Figure 1. Schematic of upper tundish nozzle (UTN) in continuous casting process



Figure 2. Bubble formation stages at refractory surface showing expansion and pressure threshold (Stage 2)



Figure 3. Geometry, mesh, and boundary conditions for 1-D test cases



Figure 4. Temperature and argon viscosity profiles in radial direction



Figure 5. Pressure distributions for 1-D test cases with fixed pressure B.C.



Figure 6. Pressure distributions for 1-D test cases with fixed gas mass flow rate B.C.



(a) Photograph of underwater experiment showing bubbling locations



distribution

Figure 7. Comparison between predicted and measured gas flow rate at UTN inner surface



Figure 8. Gas velocity distribution at symmetry plane (a) Contours of velocity magnitude; (b) Velocity vectors



Figure 9. Schematic of the base case UTN geometry (dimensions in mm)



Figure 10. Computational domain, mesh and boundary condition



Figure 11. Temperature distribution across UTN wall



Figure 12. Effect of liquid steel pressure profile on gauge pressure distribution (a) Case 1: linear pressure with bubbling threshold; (b) Case 2: linear pressure (Fig. 10); (c) Case 3: constant pressure



Figure 13. Radial velocity distribution at UTN inner surface





Figure 14. Velocity near UTN bottom showing effect of boundary conditions



Figure 15. Pressure (gauge) and velocity fields for: (a) Open bottom case; (b) Perfectly sealed case



Figure 16. Effect of gas leakage on radial gas velocity distribution at UTN inner surface



Figure 17. Effects of injection pressure (gauge) on gas velocity distribution in UTN refractory (sealed bottom)



Figure 18. Effects of injection pressure (guage) and bottom sealing on gas velocity distribution at UTN inner surface



Figure 19. Effects of injection pressure (gauge) on gas flow rate



Figure 20 Effects of permeability on radial velocity distribution



Figure 21 Effects of permeability on gas mass flow rate

2.7. References:

- B.G. Thomas: "Chapter 5. Modeling of Continuous Casting", *Making, Shaping and Treating of Steel*, 11th Edition, Vol. 5, Casting Volume, A. Cramb, ed., The AISE Steel Foundation, Pittsburgh, PA, 2003, pp. 1-24.
- [2] D. Hershey, B.G. Thomas and F.M. Najjar: "Turbulent Flow through Bifurcated Nozzles", *International Journal for Numerical Methods in Fluids*, 1993, vol. 17 (1), pp. 23-47.
- [3] Y. Xie, S. Orsten and F. Oeters: "Behavior of Bubbles at Gas Blowing into Liquid Wood's Metal", *ISIJ International*, 1992, vol. 32(1), pp. 66-75.
- [4] S.T. Johansen, F. Boysan and W.H. Ayers: "Mathematical modelling of bubble driven flows in metallurgical processes", *Applied Scientific Research*, 1987, vol. 44, pp. 197-207.
- [5] D. Mazumdar and R.I.L. Guthrie: "The Physical and Mathematical Modelling of Gas Stirred Ladle System", *ISIJ International*, 1995, vol. 35(1), pp. 1-20.
- [6] B. Li, H. Yin, C.Q. Zhou and F. Tsukihashi: "Modeling of three-phase flows and behavior of slag/steel interface in an argon gas stirred ladle", *ISIJ International*, 2008, vol. 48(12), pp. 1704-1711.
- [7] L. Wang, H. Lee and P. Hayes: "A new approach to molten steel refining using fine gas bubbles", *ISIJ International*, 1996, vol. 36(1), pp. 17-24.
- [8] A. Vargas-Zamora, R.D. Morales, M. Diaz-Cruz, J. Palafox-Ramos, and J.DE J. Barreto-Sandoval: "Inertial and Buoyancy Driven Water Flows under Gas Bubbling and Thermal Stratification Conditions in a Tundish Model", *Metallurgical and Materials Trans. B.*, 2004, vol. 35B(2), pp. 247-257.

- [9] Bai and Thomas: "Turbulent flow of liquid steel and argon bubbles in slide-gate tundish nozzles, Part I: model development and validation", *Metallurgical and Materials Trans. B.*, 2001, vol. 32(2), pp. 253-267.
- [10] X. Huang and B.G. Thomas: "Modeling of Transient Flow Phenomena in Continuous Casting of Steel", *Canadian Metallurgical Quarterly*, 1998, vol. 37(304), pp. 197-212.
- [11] T. Shi and B.G. Thomas: "Effect of Gas Bubble Size on Fluid Flow in Continuous Casting Mold", Continuous Casting Consortium at University of Illinois at Urbana-Champaign, Report, 2001.
- [12] H. Yu and M. Zhu: "Numerical Simulation of the Effects of Electromagnetic Brake and Argon Gas Injection on the Three-dimensional Multiphase Flow and Heat Transfer in Slab Continuous Casting Mold", *ISIJ International*, 2008, vol. 48(5), pp. 584–591.
- [13] V. Singh, S.K. Dash, J.S. Sunitha: "Experimental Simulation and Mathematical Modeling of Air Bubble Movement in Slab Caster Mold", 2006, *ISIJ International*, vol. 46(2), pp. 210-218.
- B.G. Thomas, X. Huang, and R.C. Sussman: "Simulation of Argon Gas Flow Effects in a Continuous Slab Caster", *Metallurgical and Materials Trans. B*, 1994, vol. 25B(4), pp. 527-547.
- [15] R. Gass, a private communication, Inland Steel, 1998.
- [16] M. Iguchi, Y. Demoto, N. Sugawara and Z. Morita: "Bubble Behavior in Hg-Air Vertical Bubbling Jets in a Cylindrical Vessel", ISIJ International, 1992, vol. 32(9), pp. 998-1005.
- [17] M. Iguchi, H. Kawabata, K. Nakajima and Z. Morita: "Measurements of bubble characteristics in a molten iron bath at 1600 °C using an electroresistivity probe", *Metallurgical and Materials Trans. B.*, 1995, vol. 26B, pp. 67-74.

- [18] H. Bai and B.G. Thomas: "Bubble Formation during Horizontal Gas Injection into Downward Flowing Liquid", *Metallurgical and Materials Trans. B*, 2001, vol. 32B(6), pp. 1143-1159.
- [19] S. Ghaemi, P. Rahimi and D. Nobes: "The effect of gas-injector location on bubble formation in liquid cross flow", *Physics of Fluids*, 2010, vol. 22, pp. 043305 1-15.
- [20] N.A. Kazakis, A.A. Mouza and S.V. Paras: "Initial bubble size through a sparger Experimental study of bubble formation at metal porous spargers: Effect of liquid properties and sparger characteristics on the initial bubble size distribution", *Chemical Engineering Journal*, 2008, vol. 137, pp. 265-281.
- [21] G. Houghton, A.M. McLean, and P.D. Ritchie: "Mechanism of formation of gas bubblebeds", *Chemical Engineering Science*, 1957, vol. 7 (1–2), pp. 40–50.
- [22] B. Bowonder and R. Kumar: "Studies in bubble formation IV: bubble formation at porous discs", *Chemical Engineering Science*, 1970, vol. 25, pp. 25-32.
- [23] G. Lee, B.G. Thomas and S. Kim: "Effect of Refractory Properties on Initial Bubble Formation in Continuous-casting Nozzles", *Metal and Materials International*, 2010, vol. 16(3), pp. 501-506.
- [24] FLUENT ANSYS Inc. 2007 FLUENT 6.3-Manual (Lebanon, NH).
- [25] S. Whitaker: "The Forchheimer equation: a theoretical development", *Transport in Porous Media*, 1996, vol. 25, pp. 27-61.
- [26] S. Ergun: "Fluid flow through packed columns", *Chemical Engineering Process*, 1952, vol. 48(2), pp. 89-94.
- [27] G.M. DÄRR, U. Ludwig: "Determination of permeable porosity", MATÉRIAUX ET CONSTRUCTIONS, 1973, vol. 6(33), pp. 185-190.

- [28] R.E. Collins: "Flow of Fluids through Porous Materials", Charles R. Wilke, Consulting Editor, Reinhold Publishing Corporation, New York, Chapman & Hall, Ltd., London, 1961.
- [29] G.H. Fancheark and J.A. Lewis: "Flow of simple fluids through porous refractory", *Industrial and Engineering Chemistry*, 1933, vol. 25(10), pp. 1139-1147.
- [30] C.A. Sleicher and M.W. Rouse: "A Convenient Correlation for Heat Transfer to Constant and Variable Property Fluids in Turbulent Pipe Flow", *International Journal of Heat and Mass Transfer*, 1975, vol.18, pp. 677-683.
- [31] P. Wesseling: "Principles of Computational Fluid Dynamics", Springer-Verlag Berlin Heidelberg, 2001, ISBN 7-03-016677-9, pp. 62.
- [32] G. Kaptay, T. Matrushita, K. Mukai and T. Ohuchi: "On different modifications of the capillary model of penetration of inert liquid metals into porous refractories and their connection to the pore size distribution of the refractories", *Metallurgical and Materials Transaction B.*, 2004, vol. 35B, pp. 471-486.
- [33] L. Jimbo, A. Sharan and A.W. Cramb: "Recent measurements of surface and interfacial tensions in steels", 76th Steelmaking Conf. Prof., Dallas, TX, 1993, pp. 485-494.
- [34] R. Dawe and E. Smith: "Viscosity of Argon at High Temperatures", *Science*, 1969, vol. 163, pp. 675-676.
- [35] R. Nunnington, LWB Refractories, private communication, Nov. 6, 2006.
- [36] B. van Leer: "Towards the Ultimate Conservative Difference Scheme, V. A Second Order Sequel to Godunov's Method", *J. Com. Phys.*, 1997, vol. 135(2), pp. 229–248.
- [37] S.V. Patankar: "Numerical Heat Transfer and Fluid Flow", Hemisphere Publishing Corporation, Washington D.C., 1980.

- [38] S.R. Mathur and J.Y. Murthy: "A Pressure-based Method for Unstructured Meshes", *Numerical Heat Transfer, Part B: Fundamentals*, 1997, vol. 31(2), pp.195-215.
- [39] S. Achaya, B.R. Baliga, K. Karki, J.Y. Murthy, C. Prakash, and S.P. Vanka: "Pressurebased Finite-Volume Methods in Computational Fluid Dynamics", *Journal of Heat Transfer*, 1996, vol. 129(7), pp. 407-424.
- [40] R. Liu and B.G. Thomas: "Model of Gas Flow through Upper Tundish Nozzle Refractory and Initial Bubble Size", *Proc. of AISTech 2012 Steelmaking Conf.*, Atlanta, GA, 2012, pp. 2235-2246.
- [41] M. Burty, M. Larrecq, C. Pusse, and Y. Zbaczyniak: "Experimental and Theoretical Analysis of Gas and Metal Flows in Submerged Entry Nozzles in Continuous Casting", *Revue de Metallurgie. Cahiers D'Informations Techniques*, 1996, vol. 93(10), pp. 287-292.

CHAPTER 3:

MEASUREMENTS OF MOLTEN STEEL SURFACE VELOCITY AND EFFECT OF STOPPER-ROD MOVEMENT ON TRANSIENT MULTIPHASE FLUID FLOW IN CONTINUOUS CASTING

3.1. Introduction

Many defects in steel products are caused by entrainment and entrapment of slag droplets and inclusion particles into the solidifying shell during continuous casting (CC) of steel (Figure 3.1a), which is directly related to fluid flow in the mold region. Many different mechanisms for slag entrainment have been proposed and investigated in past decades^[1-3], which were recently reviewed and summarized into 9 categories^[4]. These mechanisms include: mold level fluctuations, shear instability at the slag/steel interface; and are influenced by argon gas bubbles^[1-2]. Flow-related problems at the meniscus can also cause surface defects^[5]. Understanding how these mechanisms occur in response to actual flow patterns in the molten steel is crucial to reducing defect formation. Most of these proposed mechanisms are more likely to occur during transient events, due to accompanying changes in the flow pattern. For example, shear instability, i.e. the Kelvin-Helmholtz instability, occurs when velocity along the slag/steel interface exceeds a critical value^[3], which could be caused by a sudden increase of steel flow rate in the nozzle due to severe stopper-rod/slide-gate movements. Thus, understanding and avoiding defects require study of these transient events.

Several previous studies have investigated transient flow phenomena in the continuous casting mold region using computational models. Large Eddy Simulation (LES) has been applied to study transient flow during nominally steady-state flow conditions^[6-10] including particle

transport^[7], and the effect of electromagnetic forces^[9-10]. However, genuine transient events and their corresponding large-scale flow variations have received less attention due to their complex nature. Huang and Thomas^[11] developed a 3-D finite-difference model to simulate transient argon-steel two phase flow patterns in the mold, and found large scale vortex shedding phenomenon during the transition from asymmetric flow of nozzle clogging to steady symmetrical flow. Zhang et al. investigated flow during a cast startup process, utilizing a volume of fluid (VOF) model^[12]. Few studies have attempted to measure flow in the mold during transient events.

In the current work, surface velocities in the mold during transient casting conditions were measured using two different sensors, which were validated with each other, and compared with predictions of a multi-phase computational model of the flow of argon and steel in the nozzle and mold, and includes two separate submodels to predict the flow rate. The validated model was then applied to simulate a transient flow event involving multiple stopper-rod movements.

3.2. Surface velocity measurement in molten steel caster

Experimental methods are needed to monitor the real condition of flow in the mold, and to validate computational model predictions. Water models have frequently been utilized to study multi-phase flow in the continuous-casting mold^[13-15]. Many computational models have been validated using water model experiment results^[6,11,13-16]. However, it has been found^[17] that multi-phase flow behavior in an air/water system differs from that in the corresponding argon-steel system in many ways, due to differences in properties such as surface tension and contact angles leading to different bubble sizes. Similarly, oil/water systems are essentially different than

molten-slag / steel systems. Moreover, the bottom of the water model and lack of a solidifying steel shell also changes the flow^[10,18]. Thus, plant measurements are preferred over water models, especially for multi-phase flow. The liquid steel surface velocity is a key indicator of flow problems: too high a surface velocity induces excessive turbulence and shear instability at slag-steel interface and increases the possibility of slag entrainment^[11], too low a surface velocity results in excessive cooling near the meniscus regions, which may further cause hook formation, nonuniform slag consumption, and the entrapment of mold slag, inclusion particles, or bubbles, leading to various surface defects^[19]. Thus it is of great significance to find, validate, and apply methods to measure surface velocity in the mold.

3.2.1. MFC Sensor

Several different methods to measure surface velocity of liquid steel velocities have been developed and applied in previous work^[20-27]. Mass Flow Control (MFC) sensors developed by Amepa consist of a pair of electromagnetic probes imbedded behind mold walls^[20]. Liquid steel velocities were measured by computing the time delay of signals recorded by the two probes, which was caused by a change of induced electromagnetic current by flow variation as liquid metal traveled through the magnetic field^[20]. This expensive technique can accurately measure velocities only near the shell and along the top surface of the mold in regions where the steel flow is reasonably uniform^[21].

3.2.2. Karman Vortex Probe

The Karman vortex probe was developed by Iguchi et al.^[22] to measure liquid steel velocities near the meniscus based on the linear relationship between molten steel velocity and

the shedding frequency of Karman's vortex streets formed behind this cylindrical probe when it is immersed into the top surface of the mold. This method only works well in uniform flow. The probe used in this method needs a support structure above the top of the mold and requires filtering to remove noise from the signal.

Kubota et al.^[23] utilized a rod dipped into the molten steel, and measured the deflection angle of the rod and the torque acting on it. These quantities were then transformed into surface steel velocities.

3.2.3. Nailboard Method

A simple method to sample the slag layer and surface level conditions in the mold using nail boards was pioneered by Dauby et al.^[24] at LTV steel, then further developed by Thomas et al.^[25-27] to measure surface velocity. In addition to measuring instantaneous surface steel velocities in the mold and the direction of flow, the nail board method can also provide the mold level (slag-steel interface) profile across the top surface, and the thickness profile of the slag layer.

For both nail board and single nail dipping tests, nails are inserted through the top-surface powder layers into the molten steel, held for 3~5 seconds, and removed. A lump forms on the bottom of each nail, due to the solidification of the liquid steel and slag, as shown in Figure 3.1(a-b). As molten steel flows past the nail, the liquid builds up at the impinging point on the nail lump before it solidifies. The kinetic energy of the impinging stream is converted into potential energy at the stagnation point as the impinging flow rises up the nail. The liquid-steel level drops at the opposite (downstream) side of the nail lump, due to the lower pressure in the wake region. This change in level of the slag-steel interface is recorded by the shape of the solidified lump, as shown in Figure 3.1(b). By measuring the lump shape and lump height difference between the side facing the flow side and the opposite downstream side, the magnitude and direction of the surface steel velocity can be determined. A finite-element CFD model of the nail dipping test was developed by Rietow and Thomas^[18,27] to study the liquid steel flow past a nail with a liquid slag layer on top. Their steady-state three-phase model tracked the two free surfaces, by deforming the mesh to maintain cell boundaries along the liquid slag / steel interface and the slag/powder interface. This model includes the effects of interfacial tension at the slag-steel interface (assuming 1.6 N/m) and predicts the interface shape and the height difference across the nail for a given bulk velocity of the steel beneath the interface^[18,27].

3.3. Plant Measurements

In the present work, three different plant trials were conducted in the No. 1 continuous caster at ArcelorMittal Dofasco, which has a 225mm-thick mold with a bifurcated SEN with 15-deg downward ports. In trial #1, two different sensor techniques: a Sub-meniscus Velocity Control (SVC)^[28] device and single nail dipping tests, are used at the same time, to independently measure the surface velocities during a time interval with many casting speed changes. The 25-mm diameter SVC probe was inserted at a depth of 100 mm below the meniscus at the quarter point of the mold. The nail-dipping approach has since been used extensively to study the mold flow pattern in ArcelorMittal Dofasco.^[29]

Next, trial #2 is a campaign of three heats cast under controlled conditions using only the validated nail-dipping method to quantify liquid steel surface velocities, in order to validate the multi-phase CFD model. Casting speed is varied with the same argon flow rate in order to

quantify how gas fraction changes the flow pattern, which can be determined experimentally by recording the changing direction and magnitude of the surface flow velocity.

Finally, Trial #3 is a transient event with carefully recorded time-histories of the stopperrod position, the mold level measured using an eddy current sensor at mold quarter point, the casting speed, and argon gas flow rate. Sliver defects were detected on coils with the aid of downstream feedback from the Automated Surface Intelligence SystemTM (ASISTM)^[30]. An SEM image of the defects obtained from the coil sample chosen for this study are pictured in Figure 3.2. Their compositions indicate entrapped mold flux and alumina particles. Next, the locations of defects formation on slabs were calculated from the entrapment depths in the coil, knowing the defects locations along the coil length and the thickness reduction ratios between the coils and slabs. This particular defect corresponds to a shell thickness of 3.2mm in the 218mm-thick slab. This suggests that the defect chosen in this work was formed by the entrapment of inclusion particles in the mold, about 27mm below the meniscus. The time of particle entrapment was calculated from the recorded casting speed and cast length data, which defines the transient event of trial #3. The process conditions and sensing methods used in measurements for all three trials are listed in Table I.

3.4. Validation and Calibration of Nail Dipping Method

To interpret the nail-board measurements, a new equation to quantify the liquid steel surface velocity is given by Equation (3.1) based on the results from the previous computational modeling studies from Rietow and Thomas^[27], which are included in Figure 3.3. Equation (3.1) correlates the average liquid steel velocity magnitude just below the slag / steel interface with the difference in height of opposite sides of the solidified lump and with the lump diameter.

$$V_s = 0.624 \phi_{lump}^{-0.696} h_{lump}^{0.567} \tag{3.1}$$

where V_s is steel surface velocity (m/s), ϕ_{lump} is lump diameter (mm), and h_{lump} is lump height difference (mm). It is worth pointing out that the previous results by Rietow²⁷⁾ predict a maximum lump height difference, which was less than observed for lumps at high surface velocity in the current work. This suggests that Rietow's simulation results at the highest surface velocity of 0.6 m/s likely had convergence problems causing numerical error. Thus, those results (shown as hollow symbols) were excluded from the least square regression process that generated the new equation (3.1).

In trial #1, instantaneous velocities were measured by the SVC probe and recorded at a sampling frequency of 1 H_Z . A single nail with a diameter of 6 mm was inserted about 50 mm closer to the narrow face than the SVC probe for each measurement. This location for dipping the nail was chosen so that the two sensing methods were measuring velocities at almost the same location at meniscus, but also far enough apart so that the local flow around the nail did not disturb the SVC probe. Casting speed was varied greatly during trial #1 as shown in Figure 3.4(a) and the corresponding steel surface velocity histories monitored by both SVC and nail dipping are shown in Figure 3.4(b). In addition to the instantaneous SVC surface velocity measured, Figure 3.4(b) also shows a 30-second moving average velocity. The sign convention used here is that positive meniscus velocities indicate flow towards the SEN, and negative velocities indicate flow away from the SEN towards the narrow face. The locations at the top of the mold where the nail and SVC probe were inserted are also given in Figure 3.4(b). Error bars for the nail dipping test results were obtained assuming an uncertainty of 0.5 mm in measuring both the lump diameter and the lump height difference.

The SVC data and nail dipping results match closely with each other, as shown in Figure 3.4(b). Furthermore, most nail dipping measurements match the moving average of the SVC data. At a few points, the nail dipping results fall outside the moving average, but still always fall within the range of the instantaneous SVC data. Perhaps the velocities from the nail dipping measurements are slightly less than the SVC data. This might be expected, considering that the SVC probe extends to 100mm below the surface and measures an average over that range. The nail dipping test measures velocity closer to the surface, which should be lower, owing to the viscous drag effect from the slag layer.

3.5. Computational Model Description

A model system has been developed to simulate multi-phase flow evolution during transient events with actuator movements. It consists of 1) a stopper-position-based model, or a metal-level-based model to predict liquid steel flow rate inside SEN during stopper rod movements which is required as a boundary condition for the two-phase flow simulations^[31]; 2) a porous gas flow model to estimate hot argon flow rate into the liquid steel stream in the nozzle, and the resultant mean bubble size entering the nozzle, as explained elsewhere^[32]; and 3) an Eulerian-Mixture CFD model to simulate argon-steel two-phase flow in the nozzle and mold region, and a pressure-based post-processing method to estimate meniscus level. These models are summarized briefly in the following 3 sections. Further details are reported elsewhere^[31-32].

3.5.1. Stopper-position-based Model of SEN Steel Flow Rate

During steady-state continuous casting, the liquid steel flow rate into the SEN equals the throughput at mold exit. During a transient event, however, steel flow rate in the SEN varies with

time, as indicated by the rapid fluctuations of the average mold level. Two different models were developed in this work to predict the liquid steel flow rate in the SEN.

Firstly, a semi-analytical model, given in Equation (3.2), is derived from Bernoulli's equation to predict flow rate based on the measured stopper rod opening position and other parameters, with the details of the derivation found elsewhere^[31]. It is validated with plant measurements, as shown in Figure 3.5(a) and (b).

$$Q_{SEN} = A_{SEN} \left(\frac{2g \left(f_{tundish} h_{tundish} - h_{sen_sub} + L_{SEN} \right)}{1 + 0.5 \left(\frac{A_{SEN}}{C_2 h_{SRO}^2} \right)^2 + \left(\frac{A_{SEN}}{C_2 h_{SRO}^2} - 1 \right)^2 + C_1 \frac{L_{SEN}}{D_{SEN}} + C_3} \right)^{0.5}$$
(3.2)

In this equation, A_{SEN} is the SEN inner bore cross-section area; h_{sen_sub} is the submergence depth of SEN, $h_{tundish}$ is the total height of the tundish; $f_{tundish}$ is the tundish weight fraction; L_{SEN} is the total length of SEN; D_{SEN} is the SEN inner bore diameter; h_{SRO} is the stopper rod opening. The three parameters in the equation, C_1 , C_2 and C_3 , are adjustable coefficients that represent different pressure head losses: C_1 is for friction, C_2 is for the stopper rod gap and C_3 is for clogging. The influence of C_2 and C_3 on the predicted relation between stopper rod position and flow rate is shown in Figure 3.5(a) and 3.5(b). The effect of argon gas injection on the pressure head loss at the stopper rod gap is accounted for in C_2 . In this work, C_2 was calibrated using the plant trial data in Figure 3.5(a), which had the same gas volume fraction as in the current transient study (10% gas). The effect of friction factor on this relation is negligible.

3.5.2. Metal-level-based Model of SEN Steel Flow Rate

A metal-level-based model was also derived to predict steel flow rate, based on an overall mass conservation of the system. Knowing the casting speed, the time variation of the liquid

steel flow rate in the SEN can be predicted from the mold level and casting speed histories recorded in the plant as follows:

$$Q_{SEN} = \frac{dh_l}{dt} \left(WT - \frac{\pi}{4} D_o^2 \right) + V_{cast} WT$$
(3.3)

In this equation, *t* represents time; V_{cast} is the casting speed; *W* is the mold width, *T* is the mold thickness; h_l is the average mold level; and D_o is the SEN outer bore diameter. A centraldifference time-discretization of the mold level position history is used to calculate dh_l/dt , based on the eddy-current level sensor measurements midway across the mold, which are assumed to be representative of the average liquid level in the mold. This model was used as validation for the predictions from the entire model system, including the stopper-position-based flow rate model.

3.5.3. Model of Gas Flow Rate and Initial Bubble Size in Nozzle

Because gas expands at high temperature, the hot argon flow rate will be ~4 to 5 times higher in the mold than measured at room temperature (STP). The size of the resulting argon bubbles depends on gas flow through the porous nozzle refractory and significantly affects steel flow in the mold. A new 3-D porous-flow model^[32] was developed and used to calculate argon gas velocity distribution inside the UTN and exiting the inner surface, taking into account the effects of gas thermal expansion, nozzle geometry, temperature-dependent gas viscosity, and interfacial tension forces at the gas pores. After solving for the temperature distribution inside the nozzle, Equation (3.4) was solved to obtain the pressure distribution in the refractory, where *T* is the local temperature in refractory, *p* is gas pressure, *R* is the gas constant and *K*_D is the permeability.

$$\nabla \cdot \left(K_{D} \nabla p\right) = -\frac{RT}{p} \left[\nabla \left(\frac{p}{RT}\right) \cdot \left(K_{D} \nabla p\right) \right]$$
(4)

Then the gas velocities were calculated from the pressure field following Darcy's law. The results are combined with an empirical equation to estimate active sites number density from Lee et al.^[33] and a semi-empirical two-stage bubble formation model from Bai and Thomas^[34] to predict initial bubble size entering the SEN. A mean bubble diameter of 2.5 mm was obtained for this work. Details on the gas porous-flow model and calculation of the initial bubble size are given elsewhere^[32-34].

3.5.4. Multiphase Fluid Flow Model

Argon-steel two-phase flow in the nozzle and mold was simulated with a transient 3-D Eulerian-Mixture model, which satisfies mass and momentum conservation of the argon-steel mixture by solving the continuity Equations (3.5) and (3.6), and one set of Unsteady Reynolds-Averaged Navier Stokes (U-RANS) Equations (3.7):

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot \left(\rho_m \mathbf{u}_m \right) = 0 \tag{3.5}$$

where
$$\mathbf{u}_m = \frac{\alpha_s \rho_s \mathbf{u}_s + \alpha_a \rho_a \mathbf{u}_a}{\rho_m}$$
, and $\rho_m = \alpha_s \rho_s + \alpha_a \rho_a$ (3.6)

$$\rho_m \left(\frac{\partial \mathbf{u}_m}{\partial t} + (\mathbf{u}_m \cdot \nabla) \mathbf{u}_m \right) = -\nabla p + \nabla \cdot \left[\mu_m \left(\nabla \mathbf{u}_m + (\nabla \mathbf{u}_m)^T \right) \right] + \rho_m \mathbf{g} + \nabla \cdot \left(\alpha_a \rho_a \mathbf{u}_{dr,a} \mathbf{u}_{dr,a} \right)$$
(3.7)

The variables α_a and α_s represent the volumetric phase fractions of argon and liquid steel respectively, which are found by solving Equation (3.8), and knowing that α_a and α_s sum to 1.

$$\frac{\partial (\alpha_a \rho_a)}{\partial t} + \nabla \cdot (\alpha_a \rho_a \mathbf{u}_m) = -\nabla \cdot (\alpha_a \rho_a \mathbf{u}_{dr,a})$$
(3.8)

Drift velocity $\mathbf{u}_{dr,a}$ on the right hand side of Equations (3.7) and (3.8) is defined in Equation (3.9).

$$\mathbf{u}_{dr,a} = \mathbf{u}_{as} - \frac{\alpha_a \rho_a}{\rho_m} \mathbf{u}_{as}, \text{ where the relative velocity is } \mathbf{u}_{as} = \mathbf{u}_a - \mathbf{u}_s$$
(3.9)

The mixture model is then closed using an algebraic slip formulation for the relative velocity \mathbf{u}_{as} assuming that local equilibrium between phases is reached over a short spatial length, as given in equation $(3.10)^{[35]}$, where the drag function f_{drag} is taken from Schiller and Naumann^[36] and d_a is the argon bubble diameter (2.5mm), calculated as discussed in Section 5.2,

$$\mathbf{u}_{as} = \frac{\left(\rho_{a} - \rho_{m}\right)d_{a}^{2}}{18\mu_{s}f_{drag}} \left(\mathbf{g} - \left(\mathbf{u}_{m} \cdot \nabla\right)\mathbf{u}_{m} - \frac{\partial\mathbf{u}_{m}}{\partial t}\right)$$
(3.10)

The standard k- ε model was applied to model turbulence in the mixture phase.

The computational domain includes the nozzle and the liquid pool in the mold region, with the solidification front interface as the domain boundary. No-slip wall boundary condition is adopted both at the shell boundary and at the mold top surface, as the sintered slag layer serves as a solid wall, with an enhanced wall treatment $(EWT)^{[37]}$ to calculate near-wall velocities. Mass and momentum sinks are imposed at the layer of computational cells next to the shell boundary, to account for the liquid steel crossing the boundary due to solidification. Similar mass and momentum sinks are applied to quantify the argon gas escaping from the top surface. For the steady-state simulations in trial #2, one quarter of the nozzle and mold were chosen as the computational domain with a mesh of ~0.23 million hexahedral cells. For the transient simulation of 30s of trial #3, the entire-mold domain was modelled with a mesh of 0.8 million mapped hexahedral cells as shown in Figure 3.6, and time step size was 0.01sec.

3.6. Investigation of Casting Condition Effects on Surface Velocity

The trial #1 and #2 plant measurements of surface velocity and the model simulations reveal insights into the critical combined effects of gas injection, casting speed and mold width on the flow pattern in the mold. Although argon gas flow rate remains constant, the gas volume fraction varies with throughput according to changes in casting speed and width. Figure 3.7 shows the measured nail surface velocity points and SVC data samples over 1s-intervals for 3 heats of steel at each mold width (1248mm in trial #1 and 983mm in trial #2)³⁸⁾. The gas volume fraction is presented in the hot condition, which is ~4-5 times larger than measured at STP, as explained in Section 5.3. Steady-state CFD simulations were also performed for three casting speeds (1.5, 1.7, and 1.9 m/min) at the same argon flow rate (6 SLPM) of trial #2 to reveal the flow patterns, which are presented in Figure 3.8.

3.6.1. Model Validation

The calculated surface velocities are compared with results of the plant nail dipping tests in Figure 3.9, in which symbols are the mean velocities from the ten nail samples for each casting speed in trial #2, and error bars indicate the standard deviation. As shown in Figure $3.9^{[31]}$, a reasonable match is obtained between the simulated surface velocities and those from nail-dipping tests, which tends to validate the model.

3.6.2. Effect of Casting Speed

For both mold widths and a fixed gas injection flow rate (6 SLPM), Figure 3.7^[38] shows that liquid steel surface velocity increases with casting speed. For a constant mold width and gas injection rate, higher casting speed has two effects: increasing mean velocity of the liquid steel at the SEN port exit and lowering the gas volume fraction. Both effects encourage higher surface

velocities. The simulated liquid steel flow patterns in Figure 3.8 are generally double-roll flow patterns, especially at high casting speed (Figure 3.8c). In Figure 3.8(a) (1.5 m/min), however, some gas rises from the SEN port exit, drags steel upward, and causes liquid surface flow away from the SEN. This could be termed a complex flow pattern, tending towards a single-roll flow pattern. A small recirculation region is found near the surface near the SEN. Figure 3.8(b) shows that increasing casting speed (1.7 m/min) decreases the size of this recirculation region. The reverse velocity away from the SEN also decreases, as shown in Figure 3.9 while on the rest of the surface, the velocity towards the SEN increases. Increasing speed to 1.9 m/min causes the recirculation region near the SEN to disappear. Surface velocity towards the SEN further increases and the effects of gas injection become negligible. It is also observed that for higher casting speeds, the vortex center of the upper roll moves closer to SEN.

Casting speed also has a great effect on the variability of the surface velocity. Figure 3.7 shows that increasing casting speed decreases the incidence of instantaneous reverse flow at the measured points midway across the mold, especially in the SVC data. This indicates increasing tendency towards a stable double-roll flow pattern. This suggests more stable flow in the mold for higher casting speeds with double-roll flow patterns.

3.6.3. Effect of Mold Width

Figure 3.7 has results at two different mold widths. At the same casting speed, surface velocities in the narrow (983mm) mold are lower than in the wide (1248mm) mold (both SVC and nail dipping). Increasing mold width causes higher throughput, which increases steel velocity exiting the SEN ports. This increase in SEN velocity is offset slightly by the increased distance for the jet to travel from SEN port exit to meniscus, which diffuses the jet momentum

more. The net effect is that surface velocity is still higher in the wider mold (1248mm) at the same casting speed. The measurements in Figure 3.7 also suggest that this effect of mold width becomes less significant at higher casting speeds (e.g. 1.9 m/min).

Increasing mold width also appears to have increasing flow stability, as the surface velocity variations are smaller for the larger width at the same speed. However, this might be caused by the change in gas fraction. It is important to mention that this finding is based on relatively narrow mold widths (1248 and 983mm). For much wider molds (e.g. 1800mm or wider), other work^[46] suggests that flow pattern and surface velocity variations increase due to increased large-scale jet instability.

3.6.4. Effect of Gas Volume Fraction

Gas fraction changes with both casting speed and mold width, so has an important influence to explain the trends presented in the previous 2 sections. Figure 3.7 shows that the measured surface velocities at mold quarter point decrease almost linearly with increasing gas volume fraction. All three sets of measurements from both trials consistently show that increasing gas volume fraction causes a transition of flow pattern from double-roll to complex flow, especially as the gas volume fraction approaches ~10%.

Simulation results in Figure 3.8 confirm and explain this observation that increasing gas volume fraction (by decreasing casting speed), tends to change the double-roll flow pattern into a complex flow pattern. Argon bubbles are carried by the liquid steel jet into the mold, and then float up to the top surface and exit the domain, as suggested by the quasi-steady gas volume fraction distribution for trial #3 presented in Figure 3.10. The buoyant gas bubbles rising near the SEN oppose the surface flow towards the SEN from the narrow face. This reverse flow alters the

double-roll flow pattern towards a complex or even single-roll flow pattern, and causes the observed drop in surface velocity with increasing gas fraction.

3.7. Transient Event Simulation Results

The system of models is next applied to simulate the transient event of trial #3 involving multiple large stopper-rod movements, that is described in Section 2 and Table I. The stopper-rod position, mold level and casting speed histories recorded in the plant database during this event are shown in Figure 3.11. A transient two-phase flow simulation was performed with the Eulerian-Mixture model, based on the stopper-position-based flow rate, after initializing with a solution at steady-state conditions (gas distribution shown in Figure 3.10).

3.7.1. SEN Inlet Liquid Steel Flow Rate History

The liquid steel flow rate in the SEN predicted by the stopper-position-based model and the metal-level-based model are compared in Figure 3.12. Note that translating the metal-levelbased results back in time by about 1.2sec (dashed line in Figure 3.12) makes the two predicted curves roughly match. This time delay is likely related to traveling waves on the mold top surface. The average mold level based on the SEN position responds instantly to flow rate changes. Flow disturbances travel across the top surface, and the measured level signal at the mold quarter point records it later. Therefore, the SEN flow rate from the stopper-position-based model is adopted for the inlet boundary condition in the transient simulation. The simulation investigates flow pattern evolution from 9955 to 9985sec, (30.0sec), which includes a major flow-rate drop around 9965sec due to declogging, by bumping of the stopper rod in an attempt to dislodge the buildup of inclusions on the stopper tip.

3.7.2. Flow Pattern Evolution

Evolution of the flow pattern in the mold simulated during the first 16.7sec of trial #3 is shown in Figure 3.13.^[40] Each frame is plotted at the center plane between mold broad faces. The initial quasi-steady state flow field (at 9955sec), shown in Figure 3.13(a), is observed to be a symmetric double-roll flow pattern, which is expected for these conditions (9.6% gas).

As the inlet liquid steel flow rate decreases, (e.g. at 9964sec), the strength of the jets decrease. Jet strength continues to decrease (9965.7sec) and then starts to recover (at 9966.3sec). Then, (at 9967.2sec), a strong burst of liquid steel shoots up towards the meniscus near the SEN, and significant disturbance of the meniscus is observed. This likely causes liquid slag droplets to become entrained into the liquid pool. This phenomenon is probably caused by the strong buoyancy force from a large amount of rising argon gas accumulated in the nozzle during the stopper-rod closing stage. Between time 9968 and 9969.2sec, the upward liquid stream towards meniscus becomes less intensive, and liquid steel jets towards the narrow faces begin to develop, and wobbling of the jets is observed. Finally, at time 9971.7sec, the jet swinging disappears, and the symmetrical quasi-steady flow pattern becomes re-established.

3.7.3. Comparison of Predicted and Measured Mold Level

The flow pattern changes caused by the stopper rod movements also affect the mold level profile and cause fluctuations of the top surface level, which can be detrimental to steel quality. In this simulation, the top surface cannot move as a wall boundary so the local mold level is predicted from a simple energy balance, converting the pressure difference into the potential energy of the level elevation head by Equation (3.11):

$$\Delta h = \frac{p - p_0}{\rho_s g} \tag{3.11}$$

where Δh is the mold level deviation, p is pressure along the top surface, and p_0 is the pressure at the reference mold level, which is taken at the mold quarter point from steady-state solution. ρ_s is the liquid steel density (7200 kg/m³), and g is gravitational acceleration (9.81m/s). Displacement of the liquid slag layer is neglected in this equation, because the entire layer was assumed to be thin enough to simply rise and fall with the steel surface profile variations. This assumption agrees with recent measurements by Cho et al^[41].

The predicted mold level during the simulated 30.0sec interval with stopper rod movements agrees reasonably well with the measured unfiltered mold level data, at the midway point between SEN and narrow face along the centerline, as shown in Figure 3.14. However, the measured mold level signal is delayed by about 2 sec^[40]. This discrepancy is explained by the inability of the simple pressure method to capture transient waves or gravity wave sloshing, since the pressure method forces an immediate response to flow rate changes. The measured response delay consists of two parts: the time needed for the average free surface level to respond to the flow rate change, and the time for the surface wave to travel to the location of measurement (around quarter mold point). Note also that there is significant asymmetry between the left and right sides of the mold, owing to chaotic turbulence.

3.7.4. Defect Formation Mechanisms

The sliver defects in the final product produced just after this transient event are related to the evolution of the two-phase flow field. The simulations show that the multiple stopper-rod movements clearly induced flow changes in the mold and level fluctuations. Different mechanisms could explain exactly how this occurred. One possible cause is that the sudden large mold level changes immediately entrapped slag particles into the solidifying shell at the meniscus. In this case, however, ~60sec passed after the mold level disturbances before the particle was entrapped. The stopper movements likely dislodged a build-up of clog material inside the SEN, releasing inclusion particles into the mold. In addition, the surface flow variations may have entrained mold slag droplets into the steel flow in the mold. These particles then circulated in the transient mold flow for some time before eventually becoming entrapped into the solidification front, leading to slivers in the rolled product.

As a consequence of this investigation into the mechanism of defect formation, this method for declogging has been abandoned in ArcelorMittal Dofasco. Instead, a stopper-rod dithering approach has been adopted as a standard practice to reduce clogging, without significantly disturbing the molten steel surface in the mold, and has shown satisfying performance⁴²⁾.

3.8. Conclusions

Three plant trials were carried out to investigate the effect of casting conditions on fluid flow in a conventional steel slab caster mold and the cause of coil defects. A system of computational models was developed, validated with the measurements, and implemented to study both steady and transient multi-phase flow in these trials. The following conclusions are drawn:

- 1. A new correlation to quantify steel surface velocity from nail dipping plant tests, based on solidified lump height difference, and lump diameter.
- 2. An SVC system can provide reliable continuous surface velocity measurements in molten steel and successfully validated the nail dipping tests in the current work.
- 3. Nail dipping is a simple, reliable, and capable method to simultaneously measure instantaneous meniscus steel velocities and flow directions at multiple locations, in addition to slag layer thickness and surface level profiles
- 4. Flow rate models, including a stopper-position-based model, has been developed, validated and used to predict the time-dependent flow rates of steel in the nozzle, which is required to provide accurate inlet conditions for transient simulations.
- 5. Surface velocity increases with increasing casting speed and/or decreasing gas volume fraction, as the flow pattern tends towards double-roll. With high gas fractions, such as caused by low casting speed, surface velocity decreases and flow reversals (flow directed away from the SEN) are more often observed.
- 6. The flow pattern becomes more complex and continuously changing with increasing gas fraction, as the reverse surface flow away from the SEN is predicted to meet flow from the narrow face. The meeting point changes with time, causing flow reversals to be recorded at the midpoint sensor with increasing likelihood as the gas fraction increases.
- 7. The effect of increasing mold width is complicated because it increases port velocity, increases travel distance, and increases throughput, which decreases gas fraction, with a net effect of increasing surface velocity and tendency towards double-roll flow in the current study.
- 8. Transient flow events, such as due to excessive stopper rod movements, cause significant disturbances of the meniscus and transient mold flow, which may entrain slag, leading later to particle entrapment into the solidifying shell and the sliver defects in the final product.

9. The model system predictions agree reasonably well with the plant measurements. This system is a useful tool to study transient flow phenomena, especially when combined with plant measurements.

3.9. Table and Figures

Trial	Mold	Mold	Casting	Sensing	SEN Depth	Gas Injection Rate
#	Width	Thickness	Speed	Methods	(mm)	(SLPM)
	(mm)	(mm)	(m/min)			
1	1248	225	1.0 - 1.9	SVC + Nail	177	1
2	983	225	1.5 - 1.9	Nail	185	0
3	1472		1.2	Eddy Current	166	

Table I. Processing Parameters for Plant Trials



Figure 3.1. Nail dipping procedure (a) Surface slag layers; (b) Nail and lump



Figure 3.2. SEM image of subsurface defects in trial #3 $^{37)}$



Figure 3.3. Curves to convert nail lump height difference into velocity magnitude at the top surface



Figure 3.4. Casting speed change in trial #1 and measured meniscus velocity (a) Casting speed change; (b) Meniscus velocity history from SVC and nail dipping.⁴⁵⁾



Figure 3.5. Effect of C_2 and C_3 on stopper-position-based model predictions of flow rate







Figure 3.7. Effect of Casting Speed, Mold Width, and Gas Volume Fraction (in hot condition) on Surface Velocity (Trials #1 and #2)



Figure 3.8. Flow patterns with different argon gas volume fractions, with varying casting speeds



Figure 3.9. Comparison between predicted and measured surface velocities from trial #2



Figure 3.10. Quasi-steady state argon volume fraction distribution (trial #3)



Figure 3.11. Recorded stopper rod position, mold level and casting speed (trial #3)³⁸⁾



Figure 3.12. Predicted SEN flow rates from stopper-position-based and metal-level-based model (for trial #3)



Figure 3.13. Transient flow patterns during stopper rod movement in trial #3 (axis distances in m)



Figure 3.14. Comparison of calculated and measured mold level (for trial #3)

3.10. References

- [1] T. Watanabe and M. Iguchi: *ISIJ Int.*, **49**(2009), 182.
- [2] W. Emling, T. Waugaman, S. Feldbauer and A. Cramb: Proc. of Steelmaking Conf., ISS, Warrendale, PA, (1994), 371.
- [3] J. Hartman and A. Cramb: Proc. of Steelmaking Conf., ISS, Warrendale, PA, (1996), 773.
- [4] L.C. Hibbeler and B.G. Thomas: Proc. of AISTech 2013 Steelmaking Conf., Pittsburgh, PA, (2013), 1215.
- [5] B.G. Thomas: Making, Shaping and Treating of Steel, 11th Edition, ed. by A. Cramb, AISE Steel Foundation, Pittsburgh, PA, (2003),14.1.
- [6] Q. Yuan, B. G. Thomas, and S. P. Vanka: Metall. and Mater. Trans. B, 35B(2004), 685.
- [7] B.G. Thomas, Q. Yuan, S. Mahmood, R. Liu, and R. Chaudhary, *Metall. and Mater. Trans. B*, 45B(2014), 22.
- [8] R. Liu, W. Ji, J. Li, H.F. Shen and B.C. Liu: Steel Res. Int., 79(2008), 50.
- [9] R. Chaudhary, C. Ji, S.P. Vanka, and B.G. Thomas: *Metall. and Mater. Trans. B*, 42(2011), 987.
- [10] R. Singh, B.G. Thomas, and S.P. Vanka: Metall. and Mater. Trans. B, 44(2013), 1201.
- [11] X. Huang and B.G. Thomas: *Canadian Metallurgical Quarterly*, **37**(1998), 197.
- [12] Y. Wang and L. Zhang: *ISIJ Int.*, **50**(2010), 1777.
- [13] B.G. Thomas, X. Huang and R.C. Sussman: *Metall. and Mater. Trans. B*, 25B(1994), 527.
- [14] Z. Wang, K. Mukai, Z. Ma, M. Nishi, H. Tsukamoto and F. Shi: ISIJ Int., 39(1999), 795.
- [15] Y. Kwon, J. Zhang and H. Lee: *ISIJ Int.*, **46**(2006), 257.
- [16] B.G. Thomas, L.J. Mika, and F.M. Najjar: *Metall. and Mater. Trans. B*, **21B**(1990), 387.

- [17] R.C. Sussman, M.T. Burns, X. Huang and B.G. Thomas: 10th Process Tech. Conf. Proc., Toronto, Ontario, (1992), 291.
- [18] B. Rietow: MS Thesis, University of Illinois at Urbana-Champaign, Urbana, (2007).
- [19] J. Sengupta, H.J. Shin, B.G. Thomas, and S.H. Kim: Acta Materialia, 54(2006), 1165.
- [20] K.U. Kohler, et al.: Proc. of Steelmaking Conf., ISS, Warrendale, (1995), 445.
- [21] B.G. Thomas, et al.: *ISIJ Int.*, **41**(2001), 1262.
- [22] M. Iguchi and Y. Terauchi: ISIJ Int., 42(2002), 939.
- [23] J. Kubota, et al.: NKK Tech. Rev., (2001), 1.
- [24] P.H. Dauby, W.H. Emling, and R. Sobolewski: Ironmaker and Steelmaker, 13(1986), 28.
- [25] R. McDavid and B.G. Thomas: *Metallurgical and Materials Transactions B*, 27B(1996),
 672.
- [26] K. Cukierski and B.G. Thomas: *Metallurgical and Materials Transactions B*, **39B**(2007), 94.
- [27] B. Rietow and B.G. Thomas: Proc. of AISTech 2008 Steelmaking Conf., Pittsburgh, PA, (2008).
- [28] J.F. Domgin, et al.: *Revue de Metallurgie-CIT*, **10**(2005), 703.
- [29] J. Sengupta et.al.: Proc. of AISTech 2011 Steelmaking Conf., Indianapolis, IN, (2011), 1697.
- [30] E.M. Dillon, M. Sohm and D. Peters: Proc. of AISTech Steelmaking Conf., St. Louis, MO, (2008), 523.
- [31] R. Liu, J. Sengupta, D. Crosbie, M.M. Yavuz and B.G. Thomas: Proc. of AISTech Steelmaking Conf., Indianapolis, IN, (2011), 1619.
- [32] R. Liu and B.G. Thomas: Proc. of AISTech Steelmaking Conf., Atlanta, (2012), 2235.

- [33] G. Lee, B.G. Thomas and S. Kim: Met. Mater. Int., 16(2010), 501.
- [34] H. Bai and B.G. Thomas: Metall. Mater. Trans. B., 32B(2001), 1143.
- [35] ANSYS Inc. 2007 FLUENT 6.3-Manual (Lebanon, NH)
- [36] L. Schiller and Z. Naumann: Z. Ver. Deutsch. Ing., 77(1935), 318.
- [37] U. Goldberg, O. Peroomian and S. Chakravarthy: Journal of Fluids Eng., 120(1998), 457.
- [38] R. Liu, J. Sengupta, D. Crosbie, S. Chung, M. Trinh and B.G. Thomas: TMS Annual Meeting, San Diego, CA, (2011), 51.
- [39] M. Brummayer, P. Gittler and J. Watzinger: 8th VAI Continuous Casting Conference Proc., Linz, Austria, (2000), 81.
- [40] R. Liu, B.G. Thomas, J. Sengupta: Modeling of Casting, Welding, and Advanced Solidification Processes (MCWASP) XIII, Styria, Austria, June 17-22, (2012).
- [41] S. Cho, S. Kim and B.G. Thomas: University of Illinois, Urbana, IL, CCC 2013 Report, (2013).
- [42] S.D. Chung, J. Sengupta and M. Afnan: Proc. of AISTech 2013 Steelmaking Conf., Pittsburgh, PA, (2013), 1557.

CHAPTER 4.

PARTICLE TRANSPORT AND DEPOSITION IN A TURBULENT SQUARE DUCT FLOW WITH AN IMPOSED MAGNETIC FIELD

4.1. Introduction

Particle transport and deposition in turbulent flows are important in various industrial applications such as cyclone separators, dust collectors, spray combustion and transport and entrapment of inclusion particles in continuous casting (CC) of steel. A large number of studies of particle motion and deposition in wall-bounded turbulent flows were performed by previous researchers through both numerical simulations^[1-9] and experiments^[10-12]. Among these, several studies of particle transport in turbulent flows in a square duct have been previously reported^[e.g. 1-6]. Winkler, Rani and Vanka^[1, 2] performed Large Eddy Simulations (LES) of particle transport in a square duct with different particle Stokes numbers and investigated the preferential concentration of the particles. They considered one-way, two-way and four-way couplings between the continuous fluid and the discrete particle phase and simulated different particle volume fractions. They observed that the particle wall-normal deposition velocity increases with particle Stokes number, and mean secondary flows cause a wavy pattern of particle deposition velocities across the duct width. Winkler, Rani and Vanka^[1] also studied preferential particle concentrations for different particle Stokes numbers. Their results show that particles accumulate in regions with high compressional strain, and regions with low swirling strength. They demonstrated that vorticity is not always an accurate measure of preferential particle concentration especially in the near wall region where vorticity is dominated by shear.

Sharma and Phares^[3,4] performed a DNS of turbulent flow in a square duct with Lagrangian particle tracking and studied the effects of particle inertia on particle dispersion and deposition on the duct side walls. They observed that higher-inertia particles tend to accumulate near the wall and mix more efficiently along the longitudinal direction, while particles with lower inertia are more likely to be sent to the near wall region by the mean secondary flow and then drift back to the main stream, which is termed as particle re-suspension. Yao et al.^[5] also investigated particle re-suspension in a turbulent square duct flow with a relatively high bulk Reynolds number of 250,000 using LES and a dynamic Sub-grid Scale (SGS) model^[13]. They found that for smaller particles (e.g. 5 µm particle diameter), particle re-suspension is dominated by drag force due to the secondary flow, while for larger particles (e.g. 500 µm particle diameter), lift force cannot be neglected. They also evaluated the effect of gravity on particle re-suspension and concluded that in their study gravity acts against particle re-suspension^[6].

Several experiments on the deposition rate of particles in wall-bounded flows have also been conducted for years^[10-12]. Initial efforts were made by Friedlander and Johnstone^[10], who studied transport and deposition rate of dust particles onto the walls of tubes. The re-entrainment of particles was reduced to a minimum by using adhesive material to keep deposited particles stuck on the walls. They also proposed a model of free-flight mechanism for particle depositions, based on experimental measurements and theoretical analysis. Liu and Agarwal^[11] studied the turbulent deposition of aerosol particles in vertical pipe flows with different particle Stokes numbers (ranging from 0.21)

to 771) for bulk Reynolds numbers of 10^4 and 5×10^4 . In their experiments, the maximum deposition rate occurred at particle Stokes number (τ_p^+) equal to 30. They found that for particles with τ_p^+ less than 10, the dimensionless particle deposition rate increases as the second power of particle response time, and as fourth power of particle size. McCoy and Hanratty^[12] examined the experimental data thoroughly from previous work, and established the relationship between the dimensionless particle deposition rate and particle response time for different particle Stokes numbers. They also performed experimental studies of droplet deposition in a horizontal annular flow, the results of which reasonably fit their empirical correlation of the deposition rate with the particle response time.

In some industrial applications, electromagnetic devices such as MHD pumps and Electro-Magnetic Brakes (EMBr) are utilized to control the fluid dynamics of magnetically conducting fluids. It is found that when the flow is turbulent, the fluctuations are selectively damped by the magnetic field to the extent that the turbulence becomes two-dimensional. Quan, Vanka and Thomas^[14-15] conducted LES simulations of the instantaneous liquid steel flow in the mold region of a continuous caster, in which inclusion particles were released from meniscus and upstream, and trajectories of particles were computed. Chaudhary et al.^[16] performed LES simulations of liquid metal flow in a scaled model of continuous casting mold and studied EMBr effects on the flow patterns. Chaudhary, Vanka and Thomas^[17] also performed DNS simulations of the turbulent flow of a magnetic-conducting fluid in a square duct with imposed magnetic fields. The modification of the mean flow as well as near-wall turbulence by the imposed magnetic field is presented in this detailed study^[17]. The modified turbulence field

influences the mixing, particle transport and heat transfer to the walls. Since the secondary flow significantly affects the pattern of particle deposition on square duct walls^[1-6], and the applied magnetic field has a significant influence on the flow field, the particle dispersion and deposition in turn are also affected. Thus, study of particle behavior in turbulent flows with the effect of imposed magnetic field is of importance both fundamentally and practically.

In this work, we study particle dispersion and deposition in turbulent flow in a square duct at $Re_r=360$ with a magnetic field using DNS of the continuous fluid and a Lagrangian particle-tracking scheme. A pressure-based finite volume approach implemented on a Graphics Processing Unit (GPU) was used and particles with five different response times were considered. Particle dispersion, deposition locations, deposition velocities, and deposition rates for the different particle Stokes numbers were analyzed and compared with similar quantities without MHD effects. The deposition rates of particles in the non-MHD flow were confirmed to be in agreement with previous work^[2], thus validating the particle module in the code. We observe that the particle deposition rate increases with particle response time (with particle Stokes number ranging from 0.1 to 15), for both MHD and non-MHD cases. However, deposition rates at duct walls parallel to the imposed magnetic field are found to be higher than that at walls perpendicular to the magnetic field.

4.2. Numerical Formulation

Three sets of coupled equations are solved to describe the three different aspects of physics in this problem: the continuity and Navier-Stokes equations for the turbulent fluid flow, equations for the electric field, and equations for particle dynamics. For the continuous phase, the unsteady, incompressible three-dimensional continuity and momentum equations given by

$$\nabla \cdot \mathbf{u}_f = 0 \tag{4.1}$$

$$\frac{\partial \mathbf{u}_f}{\partial t} + \nabla \cdot \left(\mathbf{u}_f \mathbf{u}_f \right) = -\nabla p + \frac{1}{\operatorname{Re}_{\tau}} \nabla^2 \mathbf{u}_f + \frac{\mathbf{f}}{\rho_f}$$
(4.2)

are solved.

The source term **f** in equation (4.2) represents the Lorentz force, which is calculated by equation (4.3) with local current, **J**, and imposed magnetic field, **B**₀. The electric current in equation (4.3) is obtained from an electric potential and the instantaneous flow field, and the electric potential field is obtained by solving a Poisson equation given by equation (4.5).

$$\mathbf{f} = \mathbf{J} \times \mathbf{B}_0 \tag{4.3}$$

$$\mathbf{J} = \boldsymbol{\sigma} \Big(-\nabla \boldsymbol{\phi} + \mathbf{u}_f \times \mathbf{B}_0 \Big) \tag{4.4}$$

$$\nabla^2 \boldsymbol{\phi} = \nabla \cdot \left(\mathbf{u}_f \times \mathbf{B}_0 \right) \tag{4.5}$$

Calculations of particle dynamics are commonly based on the formulation by Maxey and Riley^[19] for the forces acting on a rigid sphere in a nonuniform flow. These include the drag force, lift force, gravitational force, pressure and stress gradient forces, Basset history force, and added-mass force. Elghobashi and Truesdell^[8] showed that for heavy particles (particles with high particle to fluid density ratio), only the drag force, lift force and Basset history force are important for particle transport. However, they also pointed out that the Basset history force due to fluid acceleration is usually an order of magnitude smaller than the drag force. Thus in current study, only the drag force and the lift force are taken into consideration, and other forces are neglected. Particle trajectories are integrated from instantaneous particle velocities in the flow field via equation (4.6), and particle velocities are computed by solving the force balance equation shown in equation (4.7) below:

$$\frac{d\mathbf{x}_p}{dt} = \mathbf{u}_p \tag{4.6}$$

$$m_p \frac{d\mathbf{u}_p}{dt} = \mathbf{F}_D + \mathbf{F}_L \tag{4.7}$$

The two terms on the right hand side of equation (4.7) are the drag force and the lift force. The drag force is calculated by equation (4.8), where the drag coefficient is calculated via a correlation^[20] with particle Reynolds number given by equation (4.9).

$$\mathbf{F}_{D} = \frac{\pi}{8} \rho_{f} d_{p}^{2} C_{D} \left| \mathbf{u}_{f} - \mathbf{u}_{p} \right| \left(\mathbf{u}_{f} - \mathbf{u}_{p} \right)$$
(4.8)

$$C_{D} = \frac{24}{\text{Re}_{p}} \left(1 + 0.15 \,\text{Re}_{p}^{0.687} \right), \text{ Re}_{p} = \frac{\rho_{f} d_{p} \left| \mathbf{u}_{f} - \mathbf{u}_{p} \right|}{\mu_{f}}$$
(4.9)

The lift force is calculated by the relation proposed by Saffman^[21] given by equation (4.10):

$$\mathbf{F}_{L} = 1.61 d_{p}^{2} \left(\rho_{f} \mu_{f} \right)^{1/2} \left| \nabla \times \mathbf{u}_{f} \right|^{-1/2} \left[\left(\mathbf{u}_{f} - \mathbf{u}_{p} \right) \times \left(\nabla \times \mathbf{u}_{f} \right) \right]$$
(4.10)

The particle response time is defined in equation (4.11), which reflects the time needed for a particle to accelerate from stationary state to about 63% of surrounding fluid velocity.

$$\tau_p = \frac{\rho_p d_p^2}{18\mu_f} \tag{4.11}$$

The particle Stokes number is defined as the dimensionless particle response time in wall units, as shown in equation (4.12) below:

$$\tau_p^+ = \frac{\rho_f u_\tau^2 \tau_p}{\mu_f} \tag{4.12}$$

The computational domain considered here has dimensions $L_x \times L_y \times L_z = 8 \times 1 \times 1$, in the streamwise (*x*-axis), parallel (*y*-axis) and transverse (*z*-axis) directions respectively. The imposed magnetic field is placed along the *z*-axis, pointing from the bottom wall to the top wall of the square duct, as shown in Figure 4.1.

The fluid flow equations are solved with periodic boundary condition in the sreamwise direction and no-slip conditions on the four side walls. For the electric potential, insulating duct walls are assumed and Neumann boundary conditions are prescribed. Thus, at walls y=0 and y=1 (parallel to magnetic field),

$$u_x = 0, \ u_y = 0, \ u_z = 0, \ \frac{\partial \phi}{\partial y} = 0$$
 (4.13)

and at walls z=0 and z=1 (perpendicular to magnetic field),

$$u_x = 0, \ u_y = 0, \ u_z = 0, \ \frac{\partial \phi}{\partial z} = 0.$$
 (4.14)

In previous studies^[1-6], streamwise periodic boundary conditions on particle positions were commonly used to study particle transport in the square duct. This treatment of particle boundary condition has two issues. First, because of periodic recycling, all particles that were originally injected in the domain will eventually deposit on the side walls. Thus the particle volume fraction in the domain keeps decreasing with time, as happens in a very long duct. However, for parametric studies, it is desirable to keep a nearly constant particle volume fraction. Second, in studying particle dispersion along the longitudinal direction of the square duct, particles distributed at different locations initially in the domain will travel at different streamwise velocities, and

particles near the walls will have much lower streamwise velocities compared with particles in the core region. The number of "duct lengths" has to be counted for each particle in order to study particle dispersion and deposition along the longitudinal direction.

In this work, we therefore followed a different practice of continuously injecting particles at the inlet (x=0) equal to the sum of particles depositing on the walls and exiting the domain. The initial locations of particles at domain inlet are randomly distributed, and the initial particle velocities are set to be equal to the local fluid velocity. When a particle travels out of the computational domain, it is not recycled back to the domain (as in the case of periodic boundary condition), but leaves the domain forever. At side walls particle boundary conditions are set as completely absorbing. Once the distance between the center of particle and the wall is less than one particle radius, the particle is considered to be deposited on the duct wall.

The coupled equations of fluid flow and MHD are discretized with 80×80x512 cells on a stretched Cartesian mesh with a stretching ratio of 1.01 from duct walls towards duct centerlines. The convection and diffusion terms in the momentum equations are discretized using a second-order Adams-Bashforth scheme and a fractional step method is used to project the pressure field to a divergence-free space. The resultant pressure Poisson equation is solved using a geometric multigrid technique with red-black successive over-relaxation (SOR) scheme. The equation for the electric potential is solved with the same algorithm^[16,17].

Particle positions are obtained using the trapezoidal rule, and the particle velocity is integrated using a 4th order Runge-Kutta method. Fluid velocities at particle locations

are interpolated using the 3-D Lagrange cubic interpolation function from a $3 \times 3 \times 3$ cell block surrounding the particle, as shown in equation (4.15), and the Lagrange multipliers are defined in equation (4.16).

$$\varphi_p = \sum_{n=1}^{27} L_x^{ni} L_y^{nj} L_z^{nk} \varphi_n$$
(4.15)

$$L_x^{ni} = \prod_{i=1,ni\neq i}^3 \frac{\left(x_p - x_i\right)}{\left(x_{ni} - x_i\right)} , \ L_y^{nj} = \prod_{j=1,nj\neq j}^3 \frac{\left(y_p - y_j\right)}{\left(y_{nj} - y_j\right)} , \ L_z^{nk} = \prod_{k=1,nk\neq k}^3 \frac{\left(z_p - z_k\right)}{\left(z_{nk} - z_k\right)}$$
(4.16)

The discretized set of flow equations, electrical potential equation and the particle transport equations are together solved on a Tesla C2075 GPU chip. Each computational cell is mapped to a GPU thread, and the flow and MHD equations (where applicable) were solved in time without the particle phase. The details of solving the pressure Poisson equation (PPE) and electric potential equation using a red-black Gauss-Seidel iterative solver with geometric multigrid technique on GPU were previously addressed by Shinn [18] and Chaudhary et al.[17] For particle simulations, each particle is assigned to an individual GPU thread and computational cells containing the particles are searched based on the updated particle positions from the previous time step. The fluid velocities are interpolated onto the particle locations following equations (4.13) and (4.14), and slip velocities and drag and lift forces are calculated in parallel for all the particles. The velocity and trajectories of the particles are updated in parallel following equations (4.6) and (4.7).

The flow Reynolds number based on the friction velocity (Re_{τ}) was 360 and in the MHD simulation, the Hartmann number was 21.2, for which the flow was still turbulent and not fully laminarized. The corresponding bulk Reynolds number was around 5000, and periodic boundary conditions were used in the streamwise direction. A 80×80 non-

uniform finite volume grid in the cross-section and 512 uniform finite volumes in the streamwise direction with $\Delta x^+=5.6$ were used. In order to demonstrate the adequacy of the mesh resolution, mean streamwise velocity profiles along both horizontal and vertical duct bisectors are plotted in Figure 4.2 and compared with those of Chaudhary et al.^[17] for a mesh of $128 \times 128 \times 512$ cells and same flow parameters (Re_r=360, Ha=21.2). As shown in Figure 4.2, very good agreement has been obtained between the two sets of results, which suggests that the results are grid-independent. It can be seen that mean velocity profile along the vertical bisector of the square duct is less flattened compared with that along the horizontal duct bisector, as a result of turbulence suppression close to the top and bottom duct walls, which are perpendicular to the imposed magnetic field. To initiate the computations, a laminar velocity profile with imposed perturbations for the first 1500 time steps was prescribed. A stationary state of the continuous flow was first obtained before particles were injected at the inlet plane. A total of 3,000,000 time steps were computed to get good stationarity of the flow, which was assessed by the attainment of a time invariant time-averaged velocity field. Table 1 gives the details of the particles released identically in the MHD and non-MHD simulations. Particle dispersion and deposition are a function of the non-dimensional response time (particle Stokes number), which ranged between 0.1 and 15, and was modified by varying the particle diameter. The fluid density was set to unity, and its dynamic viscosity was set to 0.00264. The particle to fluid density ratio was set to a value of 1000. Larger particles respond less to the instantaneous flows and concentrate differently from the smaller particles. For dilute particle loadings, the one way coupling is a valid assumption. Although, Elghobashi^[8] recommends a criterion of 10^{-6} below which the one-way coupling is definitely applicable, this criterion makes the particle loading very dilute to get a large enough sampling size. Winkler et al.^[2] performed simulations of particle laden flow in a square duct (without MHD effects) with a similar computational algorithm as in the current work and found that the particle deposition patterns with one-way coupling were in close agreement with those of a four-way coupling calculation for particle volume fractions around 10^{-4} and particle Stokes numbers between 0.1 and 30. Based on this observation, we have currently used a maximum volume fraction of the particle phase to be 10^{-4} .

4.3.Results and Discussion

4.3.1. Continuous Flow Fields

In a turbulent flow, particle transport is significantly more complex than in a laminar steady flow. Because of the turbulent fluctuating flow and the vortical structures, at any instant in the simulation, the local velocities provide the instantaneous drag and lift forces based on positional velocity vector and the particle parameters. Hence the generation of the continuous flow field is quite important to describing particle dynamics in an accurate way. In our study, we have first generated a statistically stationary flow by integrating the discrete equations until the ensemble-averaged flow field reached a stationary state. Particles of desired size were then released while the continuous flow fields for a MHD duct flow are significantly different from those of a non-MHD case, it is expected that the dispersion and deposition characteristics will be quite different for the two cases and also will depend in a complex way on the particle Stokes number.

For a given continuous flow, the dispersion and deposition characteristics will also depend on the manner by which the particles are injected in the duct. In several previous studies (such as [1-6]), the particle positions were "recycled" from the duct exit to the entrance of the periodic duct, thus simulating a long fully-developed duct flow. However, in such a case, the particle statistics will depend on the length of the integration time (equivalent to duct length). Also, only an average behavior over the duct length and integration time is obtained. Instead, in the present study, we inject the particles continuously at the inlet at randomly selected cross-sectional positions. The number of particles injected at any time equals the sum of number deposited and the number of particles exiting the outlet, so that the total number of particles in the computational domain remains the same. Thus the statistics of the deposition correspond to a fixed length of duct with averaging only in time. The particle deposition will therefore be a function of duct axial length and fully reflects the physics of the particle deposition.

Figure 4.3(a) and 3(b) show representative instantaneous velocity fields at crossstream planes for the non-MHD and MHD cases respectively. The contours correspond to streamwise velocity, while the vectors indicate the cross-stream secondary flow. The instantaneous secondary velocities are typically 8-10 percent of the local instantaneous streamwise velocities, but when averaged in time reduce to much smaller time-averaged values. As a result, modeling the particle dynamics using Reynolds-averaged turbulent fields is quite difficult since the extraction of the instantaneous flow "backwards" is not possible. DNS provides the time instantaneous flow fields, although being limited to low Reynolds numbers. It can be seen that the MHD and non-MHD instantaneous flow fields are considerably different. This is because of the additional $(\mathbf{J} \times \mathbf{B}_0)$ force acting in the x- and y- directions. The additional force in the streamwise direction suppresses streamwise turbulence intensity while the x- direction force acts to modify the cross-stream turbulence. Since Chaudhary et al.^[17] have already reported the rms statistics as well as the turbulent kinetic energy budgets, we limit our discussion to the particle dynamics only and provide the flow fields for completeness. Further, due to the assumption of the one-way coupling, the continuous flow field remains the same (statistically) as the noparticle case previously reported by Chaudhary et al.^[17].

Figures 4.4(a) and 4(b) show the ensemble–averaged secondary flows and the contours of streamwise velocity. For the non-MHD case, the flow is symmetric about one eighth of the duct taking into account proper reflections. However, with MHD, there is only symmetry about one fourth of the duct with reflection. The mean secondary flow eddies in the MHD case are stronger at the top and bottom walls (perpendicular to the magnetic field) and smaller towards the center of the walls and the duct corners. The mean streamwise velocity distribution is also modified by this drifting velocity, shrinking along the magnetic field direction. Eddies along the side-walls parallel to the magnetic field are weakened but get closer to corners. This modification of mean secondary flow pattern will significantly change the pattern of particle deposition. While the mean flow field does not adequately characterize the particle dynamics, it is the one that is often predicted in engineering simulations and supplemented with models for stochastic dispersion. However, such predictions can be grossly in error because of the complex turbulent fields.

4.3.2. Particle Dispersion

The deposition of particles on duct walls is directly a result of how the particles are transported by the turbulent eddies. The focus of this paper is to study the particle dynamics in turbulent duct flow with a magnetic field and to compare it with that in a non-MHD flow. The instantaneous flow is used to transport the particles injected at the duct inlet. Since only one-way coupling is considered in this study, the continuous phase flow velocities are not altered by particles.

It is well-known from previous studies that particles preferentially concentrate in regions of low vorticity and high strain rate^[9]. There are several ways to illustrate this effect. By overlapping the particle positions and the instantaneous velocity field, we can study how particles preferentially accumulate in certain regions versus others. It is also known that particles collect in regions of low swirling strength and high strain. Both of these can be attained from the fluid velocity gradient tensor used to identify vortices^[23]. Since vorticity can arise from both swirl and shear, it is not a clearly defined quantity for particle accumulation. The swirl strength is also related to the magnitude of the centrifugal effect experienced by the particles. Another useful quantity to characterize the effects of flow structures on particle dispersion was proposed by Maxey^[24] to be $\nabla \cdot \mathbf{u}_p$ which acts like a source term in the particle equation of motion. It is given by

$$\nabla \cdot \mathbf{u}_p \approx -\tau_p \nabla \mathbf{u}_f : \nabla \mathbf{u}_f \tag{4.15}$$

when only the drag force is included. Thus positive double dot product of the fluid velocity gradient tensor gives rise to particle accumulation and negative double dot product shows particle voids. Small particle response times show less preferential concentration.

Winkler et al.^[1, 2] have previously studied particle accumulation by turbulence driven secondary flows in a square duct without a magnetic field. However, their method recycled the particles from outlet to the inlet, and also performed streamwise averaging of particle statistics. In the current study, particles are continuously injected with random locations at the inlet. Hence the gradual preferential concentration of particles can be more clearly seen in our study.

Figure 4.5 shows the positions of particles at one instant in time for the MHD and non-MHD cases for two particle Stokes numbers. The cross-stream secondary velocities are also shown. It can be seen that the particles, as expected, accumulate in saddle regions of cross-stream vortices. The smaller particles (lower Stokes number) are less concentrated because of their faster adjustment to the flow, as also seen in equation (4.15). Figure 4.6 shows the contours of $\nabla \cdot \mathbf{u}_{p}$ obtained from the continuous flow field using the relation in equation (4.15). The particles will concentrate in regions of negative $\nabla \cdot \mathbf{u}_{p}$ and will be pushed out of positive $\nabla \cdot \mathbf{u}_{p}$ regions. The four instantaneous contours of $\nabla \cdot \mathbf{u}_{p}$ and the particle locations (dots) do indicate this correlation well. We notice that in most places the particles are concentrated where $\nabla \cdot \mathbf{u}_{p}$ is negative. There are essentially no particles in regions of positive $\nabla \cdot \mathbf{u}_{p}$ (red regions in color version). However, the correlation is not perfect because of the time varying $\nabla \cdot \mathbf{u}_p$ and the particle positions. Figures 4.7 and 4.8 show the preferential concentration of particles in planes parallel to the walls. Since the flow fields parallel to one set of walls are different from the other set, the particle distributions are also different. In both Figure 4.7 and 4.8, it is seen that

particles tend to accumulate in regions with low-velocity streaks. Comparison of the streamwise velocity streaks in Figure 4.7(a) and 4.7(b) reveals a thinner and more elongated streaky structure in the MHD flow shown in Figure 4.7(b) as reported previously in [17], which in turn lead to a more concentrated particle dispersion in the MHD flow for St=5 particles. However, particles with a larger response time (St=15) are more randomly distributed and not sensitive to this difference in the streak structures between the MHD and non-MHD case, as is observed in Figure 4.8(a) and 4.8(b).

4.3.3. Particle Deposition

4.3.3.1. Deposition locations

In this section, we present the particle deposition probability density distributions and particle preferential deposition locations. In order to numerically obtain a distribution of deposited particles in the streamwise direction, each duct wall is divided into 200 bins along the streamwise direction and the number of particles in each of the bins is counted which is then divided by total number of particles to calculate the local deposition probability.

Total deposition of particles on duct walls is one of the key aspects of particle transport. Deposition of particles is influenced in a complex way by the local instantaneous flow velocities and the particle Stokes number as well as its instantaneous velocity from previous time step. In the case of one-way coupling the continuous flow field is not modified by the particles, thus the interaction is somewhat less complex. As the local flow fields with and without MHD are quite different, the deposition rates and patterns are quite different in the case of MHD and non-MHD duct flows. The probability distribution function of particle deposition along the streamwise direction is shown in Figure 4.9, for the five particle Stokes numbers with and without MHD effects. An important observation from this study is that deposition probability of particles is decreased significantly by the imposed magnetic field, for all the particle Stokes numbers. The deposition rate also decreases with the streamwise location along the square duct in all five cases. Initially, an increase of the slope close to the domain inlet is also observed. This might be a result of the random introduction of particles locations with local fluid velocity at domain entrance. This effect of initial condition vanishes after a small distance (x=0.5) along streamwise direction. The deposition probability increases significantly with particle Stokes number. As the Stokes number increases from 0.1 to 15, the pdf increases by approximately 3 orders of magnitudes for both MHD and non-MHD cases. As discussed by Brooke et al.^[22], two different mechanisms are responsible for the particle deposition in wall bounded turbulent flows: particle inertia and turbulent diffusion. For particles with higher inertia, they have the energy to penetrate the boundary layer and get deposited quickly, as described by the "free-flight" model^[10, 21]. For low-inertia particles, the dominant way of deposition is through the turbulent diffusion. As particle Stokes number increases, effect of particle inertia becomes more important in determining the deposition of particles.

The distribution of particle deposition pdf along the y- and z- directions on the duct walls are plotted in Figure 4.10(a) and 4.10(b). These are summed over the length of the duct. Note that pdf distributions of particles deposited on opposite walls are expected to be statistically identical. Thus the particle deposition pdf presented in Figure 4.10 are averaged over opposite walls. Figure 4.10(a) shows the particle deposition pdf on side

walls parallel to the magnetic field (y=0 and y=1). It is observed that for all four cases, the deposited particles preferentially concentrate in regions close to the corner and in the central region of the wall. A wavy pattern for the preferential deposition locations is observed. For the non-MHD case, particle deposition pdf patterns on side walls parallel and perpendicular to the magnetic field reflect each other, as expected. Higher particle response time (St=15) tends to make the deposition patterns less preferential.

In the MHD case (Figure 4.10b), particles of smaller Stokes number (St=5) have a higher variation in the pdfs on both parallel and perpendicular walls (to the magnetic field). Particles deposited on duct walls parallel to the magnetic field exhibit a more wavy distribution compared to the non-MHD case, while for walls perpendicular to magnetic field, particle depositions show peaks near the two corners, with low particle deposition rate in the central region (between 0.2 and 0.8) of the walls. The particle deposition pdf in the central region is only 5% of that in the non-MHD case.

For the MHD case with a higher particle response time (St=15), the pdf distributions between parallel (to magnetic field) and perpendicular walls have less variation compared to the cases with smaller particle response time. Particle distributions near duct corner regions are similar for parallel and perpendicular walls. However, the shape of the pdf curve in the central region has an inflexion point. The total deposition rate on walls parallel to the magnetic field is higher than that on the perpendicular walls.

The effect of periodic cycling of particles versus random injection at inlet is examined in Figure 4.11 by comparing particle deposition patterns of current simulation and calculation of Winkler et al.^[2] for similar particle Stokes number. A reasonable match of particle deposition pdf in the middle region of the duct walls is seen between the two particle injection methods, with some difference at the corners. The current method of continuous random particle injection at duct entrance leads to a local peak in the particle deposition near the duct corner, while the method with periodic particle cycling shows a relatively flat profile.

Figures 4.12(a-b) and Figures 4.13(a-b) show the spatial distributions of particles deposited on the duct walls in both MHD and non-MHD cases for the lower particle response time (St=5). These clearly show that the deposited particle distribution on parallel walls and perpendicular walls for non-MHD square duct flows are similar. However, for the MHD flow the particle deposition on walls perpendicular to magnetic field are significantly different from the non-MHD flow case. It can be seen that in the presence of the magnetic field very few particles deposit in the central region of the wall. The particles on these walls deposit preferentially in regions close to the duct corners. Particle deposition on walls parallel to magnetic field is seen to exhibit a streaky pattern in both MHD and non-MHD cases. The location of the deposited particles in Figure 4.12(a), 4.12(b) and Figure 4.13(a) and 4.13(b) are an aggregate over a time span of Δt^+ =70.

4.3.3.2. Wall-normal and streamwise deposition velocities

Particle deposition velocities are an important factor for pipe clogging and erosion. Figure 4.14 shows the wall-normal velocity distributions of the depositing particles with two different particle Stokes numbers (St=5 and 15), for both MHD and non-MHD cases. Wall-normal deposition velocities of the St=5 and St=15 particles are shown in Figure 4.14 (a), with wall-normal fluid velocities at y^+ =3.67 plotted for

comparison. It is seen that in the non-MHD case as shown in Figure 4.14 (a), the particle deposition velocities in the wall-normal direction for both Stokes numbers are higher than those of the continuous phase in the viscous sublayer ($y^+=3.67$). Also the wall-normal velocities of the deposited particles with larger response time (St=15) on walls both parallel and perpendicular to the magnetic field (pointing towards positive z- axis) are about 4 times higher than those of the particles with the smaller response time (St=5). It is also observed that for both particle Stokes numbers presented here (St=5 and 15), a wavy profile of the particles wall-normal deposition velocities with valleys and peaks forms in the non-MHD case, which matches with previous results by Winkler^[1]. The locations of the two secondary peaks for the particle wall-normal deposition velocities near the duct corners change slightly with the particle response time: larger particle response time (St=15) tends to move the secondary peak locations towards the duct center (~13% of the duct width), while smaller particle response time (St=5) keeps the secondary peak locations closer (~9% of the duct width) to the peak locations of the mean wall-normal velocities in the viscous sublayer (~5% of the duct width in the plane of $y^+=3.67$). Thus it is suggested that for the non-MHD case from the comparison between the two particle Stokes numbers presented here (St=5 and 15), wall-normal particle deposition velocities on walls both parallel and perpendicular to the magnetic field increase with particle response time by ~4 times (from St=5 to St=15). A wallnormal velocity peak exists around the duct center due to the lift force pointing towards the wall induced by relatively large local streamwise velocity gradient, as reported previously by Winkler et al.^[1].

Wall-normal velocity distributions of depositing particles in the MHD case on walls parallel and perpendicular to the magnetic field are presented in Figure 4.14(b) and 4.14(c), for both particle response times (St=5 and 15). It is clearly seen that wall-normal particle deposition velocities in the MHD case are significantly altered from those in the non-MHD case. High wall-normal deposition velocity region at duct center in the non-MHD case has the lowest particle wall-normal deposition velocities in the MHD case on walls perpendicular to the magnetic field. The two secondary peaks in the wall-normal velocity profiles in the MHD case on walls perpendicular to the magnetic field stay at the same deposition locations as in the non-MHD case. However, the wall-normal particle deposition velocities at these secondary peaks on duct walls perpendicular to the magnetic field are ~30% less than those in the non-MHD case for particles with a higher response time (St=15), and around the same for the smaller particle response time (St=5). The wall-normal deposition velocities at duct center decrease by ~70% in the MHD case on walls perpendicular to the magnetic field for the larger particle response time (St=15), and almost 100% for the smaller particle response time (St=5). On walls parallel to the magnetic field, however, no secondary peaks for the particle wall-normal deposition velocities exist, leaving only the peak around duct center region, with the maximum averaged particle deposition velocity decreased by ~25% from that in the non-MHD case for St=15 particles, and ~50% for St=5 particles.

Streamwise averaged depositing velocities for both St=5 and St=15 particles adopt similar peak locations with those for the wall-normal deposition velocities as seen in Figure 4.15 (a)-(c). Mean fluid streamwise velocities at planes 3.67 wall units from the duct side walls are lower than the particle streamwise deposition velocities for St=15
particles, but higher than those for the St=5 particles in the non-MHD case, as shown in Figure 4.15(a). In the MHD case on walls perpendicular to the magnetic field, particle deposition velocities are ~40% lower than the fluid mean velocities at duct center for St=15 particles, but more than one order of magnitude lower for St=5 particles, as shown in Figure 4.15(b). Figure 4.15(c) gives the streamwise mean profile of particle deposition velocities on walls parallel to the magnetic field, together with the fluid streamwise mean velocity profile in the viscous sublayer (y^+ =3.67). The maximum streamwise deposition velocity on walls parallel to the magnetic field is decreased from ~5.6 in the non-MHD case to ~4.4 with a 21% reduction for St=15 particles, and a ~33% reduction for St=5 particles.

4.3.3.3. Deposition rate

Particle deposition rate has been defined in different ways in the literature^[3,7,21]. In this work, the deposition rate is computed following equation (4.15), where N_d is the number of particles deposited on all duct walls over a time span of Δt_d , N_p is the number of particles in the domain with a volume V and a deposition surface area A, and the deposition rate is non-dimensionalized by the friction velocity u_τ , as defined in [3]. However, it is worth pointing out that all definitions from [3, 7, 21] are equivalent to each other if only uniform-sized particles are considered.

$$V_d^+ = \frac{N_d V}{\Delta t_d N_p u_\tau A} \tag{4.15}$$

As shown in Figure 4.16, current results of the total deposition rates in the non-MHD case agree well with the previous data from one-way coupling simulations of Winkler et al.^[1,2], despite the difference in particle deposition pdf near duct corner region observed in Figure 4.11. This agreement on deposition rate between previous work and current results suggests that the particle module in current GPU code is implemented correctly, and re-enforces the pattern of particle deposition in a turbulent square duct flow, which is different than that in a pipe flow^[11] and in annular flows^[12].

It is observed that for larger particle Stokes numbers (e.g. $St \ge 5$) in both MHD and non-MHD cases, the computed deposition rates match reasonably well with experimental correlations by both $Liu^{[11]}$ and $McCoy^{[12]}$. Thus deposition of particles of higher inertia is not influenced by the turbulence driven secondary flows compared with lower-inertia particles. For particles with lower inertia (e.g. St<5), the deposition rates start to deviate from the experimental correlations for pipe flows and remain higher. This discrepancy is due to the effects of turbulence-induced secondary flows unique to non-circular ducts, which constantly send particles towards duct walls with a net effect of enhancing particle depositions.

In the MHD case, it is seen that the rate of particle deposition on walls parallel to the magnetic field is 2~5 times larger than that on walls perpendicular to the magnetic field. However, this difference reduces when the particle Stokes number increases (e.g. St>5). It is seen that for St larger than 10, the particle deposition rate on walls parallel to the magnetic field is just slightly higher (~20%) than that on the walls perpendicular to the magnetic field. The deposition rates in the non-MHD case for all the particle Stokes numbers studied in current work are usually 2~5 times higher than those in the corresponding MHD case, while this ratio also decreases as particle response time increases.

4.4.Conclusions

In this work, a DNS study on turbulent flow in a square duct with the effects of an imposed magnetic field was first performed, and then Lagrangian particle tracking was utilized to investigate particle dispersion and deposition using one-way coupling. Inclusion of magnetic field modifies the flow in the streamwise direction as well as secondary mean flows. Turbulence is suppressed with the effect of imposed magnetic field. Resultant secondary mean flow does not exhibit a symmetric pattern along the corner bi-sector any more, with the mean secondary eddies along the walls parallel to the magnetic field weakened, and the one along the walls perpendicular to magnetic field shifted towards duct core.

In both MHD and non-MHD square duct flows, particles tend to accumulate in the saddle regions between turbulent eddies, but away from the centers of the secondary vortices of the cross-flow direction. Along the streamwise direction close to the wall, particles tend to gather in regions with low velocity streaks. Pattern of particle deposition on the duct walls has been significantly altered by the imposed magnetic field, with the particle deposition rate decreased from ~7% to ~2% (e.g. for St=5). Preferential particle deposition location for no-MHD case is observed to have a wavy shape along the spanwise direction, with more particles deposited near the corner region, and in the central region of the wall. Similar deposition pattern is found in the MHD case at walls parallel to the direction of imposed magnetic field. However, at walls perpendicular to the magnetic field direction, the number of deposited particles decreases substantially in the central region, especially for particles with smaller Stokes number (e.g. St<5), while more particles can be found near the corners of the duct. Increasing particle Stokes

number increases particle deposition rate and particle deposition velocities in both MHD and non-MHD cases. The average streamwise velocities of depositing particles are smaller than the local averaged fluid velocities at 3.67 wall units for smaller particles (e.g. $St \le 5$), but larger for larger particles.

Results from particle deposition rate calculations suggest that overall particle deposition rates in non-MHD cases are usually 2~5 times higher than those in the corresponding MHD cases, while this ratio drops to almost equal for particles with larger response times (e.g. St>10), the deposition rates of which also follow the previous experimental correlations from pipe and annular flows. Deposition rates of particles with smaller response times in both MHD and non-MHD cases deviate from those correlations mainly due to the unique secondary flows induced by turbulence in square duct flows.

4.5.Tables and Figures

Particle Stokes	Particle	Response	Particle Volume
Number	Diameter	Time	Fraction
τ_p^+	d_p	$ au_p$	ϕ_{v}
0.1	0.000118	0.000293	5.356×10 ⁻⁸
1	0.000373	0.00293	1.694×10^{-6}
5	0.000833	0.0146	1.894×10^{-5}
10	0.00118	0.0293	5.356×10 ⁻⁵
15	0.00144	0.0439	9.840×10 ⁻⁵



Figure 4.1. Schematic of computational domain



Figure 4.2. Velocity profiles along horizontal and vertical bisectors in MHD duct flow



Figure 4.3. Instantaneous velocity field in a cross-sectional plane



(b) MHD case

Figure 4.4. Time-averaged velocity field in a cross-sectional plane



Figure 4.5. Instantaneous particle positions in a cross-sectional plane



Figure 4.6. Preferential particle concentration in a cross-sectional plane



Figure 4.7. Instantaneous streamwise velocity contours with particle positions for St=5



Figure 4.8. Instantaneous streamwise velocity contours with particle positions for St=15



Figure 4.9. Probability distribution function of particle deposition location along streamwise direction



(b) pdf on walls perpendicular to magnetic field

Figure 4.10. Probability distribution function of particle deposition location along spanwise and transverse directions



Figure 4.11. Comparison of particle deposition pdf for two particle releasing approaches



Figure 4.12. Particle deposition locations on walls parallel to the magnetic field (St=5)



Figure 4.13. Particle deposition locations on walls perpendicular to the magnetic field (St=5)



Figure 4.14. Wall-normal velocity distribution of depositing particles



Figure 4.15. Streamwise velocity distribution of depositing particles



Figure 4.16. Dimensionless particle deposition rates

4.6.Reference

- Winkler, C.M., Rani, S.L., and Vanka, S.P., 2004, "Preferential Concentration of Particles in a Fully Developed Turbulent Square Duct Flow", *International Journal of Multiphase Flow*, **30**(1), pp. 27-50.
- [2] Winkler, C.M., Rani, S.L., and Vanka, S.P., 2006, "Numerical Study of Particle Wall-deposition in a Turbulent Square Duct Flow", *Powder Technology*, **170**(1), pp. 12-25.
- [3] Phares, D.J. and Sharma, G., 2006, "A DNS Study of Aerosol Deposition in a Turbulent Square Duct Flow", *Aerosol Science and Technology*, **40**(11), pp. 1016-1024.
- [4] Phares, D.J. and Sharma, G., 2006, "Turbulent Transport of Particles in a Straight Square Duct", International Journal of Multiphase Flow, **32**, pp. 823-837.
- [5] Yao, J. and Fairweather, M., 2010, "Inertial Particle Resuspension in a Turbulent, Square Duct Flow", Physics of Fluids, 22, 03303.
- [6] Yao, J. and Fairweather, M., 2012, "Particle Deposition in Turbulent Duct Flows", Chemical Engineering Science, 84, pp.781-800.
- [7] Narayanan, C. and Lakehala, D., 2003, "Mechanisms of Particle Deposition in a Fully Developed Turbulent Open Channel Flow", Physics of Fluids, 15(3), pp.763-775.
- [8] Elghobashi, S., 1994, "On Predicting Particle-Laden Turbulent Flows", Applied Scientific Research, 52, pp. 309-329.

- [9] Kulick, J., Fessler, J. and Eaton, J., 1994, "Particle response and turbulence modification in fully developed channel flow," Journal of Fluid Mechanics, 277, pp.109–134.
- [10] Friedlander, S.K. and Johnstone, H.F., 1957, "Deposition of Suspended Particles from Turbulent Gas Streams", Industrial and Engineering Chemistry, 49(7), pp.1151-1156.
- [11] Liu, Y.H. and Agarwal, J.K., 1974, "Experimental Observation of Aerosol Deposition in Turbulent Flow", Aerosol Science, 5, pp.145-155.
- [12] McCoy, D.D. and Hanratty, T.J., 1977, "Rate of Deposition of Droplets in Annular Two-Phase Flow", International Journal of Multiphase Flow, 3, pp.319-331.
- [13] Germano, M., Piomelli, U., Moin, P., and Cabot, W.H., 1991, "A Dynamic Subgrid-Scale Eddy Viscosity Model", Physics of Fluids, 3, pp. 1760-1765.
- [14] Yuan, Q., Thomas, B.G., and Vanka, S.P., 2004, "Study of Transient Flow and Particle Transport during Continuous Casting of Steel Slabs, Part 1. Fluid Flow", Metallurgical and Materials Transactions B, 35B(4), pp. 685-702.
- [15] Yuan, Q., Thomas B.G., and Vanka S.P., 2004, "Study of Transient Flow and Particle Transport during Continuous Casting of Steel Slabs, Part 2. Particle Transport", Metallurgical and Materials Transactions B, 35B(4), pp. 703-714.
- [16] Chaudhary, R., Thomas, B.G., and Vanka, S.P., 2012, "Effect of Electromagnetic Ruler Braking (EMBr) on Transient Turbulent Flow in Continuous Slab Casting using Large Eddy Simulations," Metallurgical and Materials Transactions B, 43B (3), pp. 532-553.

- [17] Chaudhary, R., Vanka, S.P. and Thomas, B.G., 2010, "Direct Numerical Simulations of Magnetic Field Effects on Turbulent Flow in a Square Duct", Physics of Fluids, 22(7), 075102.
- [18] Shinn, A.F., Vanka, S.P. and Hwu, W.W., 2010, "Direct numerical simulation of turbulent flow in a square duct using a graphics processing unit GPU", Proceedings of the 40th AIAA Fluid Dynamics Conference, June 2010, AIAA Paper No. 2010-5029.
- [19] Maxey, M.R. and Riley, J.K., 1983, "Equation of Motion for a Small Rigid Sphere in a Nonuniform Flow," Physics of Fluids, 26, pp. 883-889.
- [20] Schiller, L., Naumann, Z., 1935, "A Drag Coefficient Correlation", Z. Ver. Deutsch. Ing., 77, pp. 318-320.
- [21] Saffman, P.G., 1965, "The Lift on a Small Sphere in a Slow Shear Flow," Journal of Fluid Mechanics, 22, pp. 385-400.
- [22] Brooke, J.W., Hanratty, T.J., and McLaughlin, J.B., 1994, "Free-Flight Mixing and Deposition of Aerosols", Physics of Fluids, 8(10), pp. 3404-3415.
- [23] Adrian, R.J., Christensen, K.T. and Liu, Z.-C., 2000, "Analysis and Interpretation of Instantaneous Turbulent Velocity Fields", Experiments in Fluids, 29, pp. 275-290.
- [24] Maxey, M. R., 1987, "The Gravitational Settling of Aerosol Particles in Homogeneous Turbulence and Random Flow Fields", Journal of Fluid Mechanics, 174, pp. 441-465.

CHAPTER 5.

COMPUTATIONAL AND EXPERIMENTAL STUDIES OF ARGON-STEEL FLOWS IN A CONTINUOUS CASTER MOLD

5.1 Introduction

Argon injection into the upper tundish nozzle (UTN) during continuous steel casting is a common practice to prevent re-oxidation and clogging (Figure 5.1).^[1] This injected gas significantly influences the liquid steel flow in both the nozzle and mold regions. The argon-steel interaction in the nozzle and in caster molds has been extensively investigated previously via both cold model experiments and numerical simulations.^[2-12] Water model experiments have been conducted to determine the different tow-phase flow regimes and liquid steel flow patterns during continuous casting.^[2-3] However, as previously pointed out by Huang and Thomas,^[4] differences in the physical properties (e.g. surface tension and density) and in the operating conditions (heat transfer process) between the air-water and argon-steel systems impose limitations in understanding of argon-steel flows from water model experiments measurements/observations. Thus it is necessary to adopt computational tools to quantitatively study argon-steel two-phase flows in caster molds.

Numerical simulations to investigate the two-phase flow dynamics during continuous casting have used different multiphase models.^[4-12] One of the major difficulties in modeling gas-liquid two-phase flows in general is to determine the multiphase flow regimes prior to the simulations, e.g. bubbly flows, slug flows, plug flows or annular flows (for flow in a duct or pipe), since different models apply to different flow regimes. Gas volume fraction is usually used

as a criterion to estimate the flow regime. In continuous casting, gas volume fraction usually varies between 0-10%, which usually falls into a bubbly flow regime for argon-steel mold flows.

For both physical and numerical modeling approaches to study argon flows in continuous casting, two parameters regarding the injected gas are of great importance to the accuracy of modeling results: argon gas volumetric flow rate entering "hot" liquid steel stream and the initial bubble size distribution. Despite their significance in understanding the multiphase flow behavior during continuous casting, these two parameters have not been studied much in previous work. Argon gas expands while traveling through the UTN refractory due to increasing local temperature as it approaches the liquid steel. The temperature-dependent gas viscosity also changes the local permeability, as discussed in Chapter 2. Huang and Thomas^[4] showed a simple equation to convert the measured "cold" gas flow rate to that in the "hot" condition, based on the ideal gas law to account for the gas thermal expansion and pressure drops, in order to estimate the hot argon gas flow rate for two-phase caster nozzle/mold simulations. In previous modeling work, this conversion has frequently been used with success.^[4,9] However, this work all has assumed that all of the gas enters the steel, so that leakage is negligible in the system, which might not be true during real casting processes. Gas could leak out of the system through the delivery system, possible cracks in the nozzle refractory, or joint openings.

Bubble size is another important parameter that affects momentum exchange between the gas and liquid phases. Shi^[7] performed parametric studies to investigate the effects of bubble size and liquid/gas flow rates on the two-phase flow fields in the mold region, using different multiphase models. Bubble size evolution was also considered in this work to incorporate the effects of bubble break-up and coalescence on the two-phase flow interactions, using a multi-sized group model (MUSIG).^[33] Insights were gained by the simulation results with a

distribution of different bubble sizes and the resultant liquid flows in the mold region. A flow pattern transition from double-roll to single-roll was indicated with increasing gas volume fraction. This work,^[7] shows that the use of the initial bubble size in general has a great influence on simulated two-phase flow patterns in nozzle and mold region, and enabled a good match with measurements. However, bubble size is still not well understood, owing to the difficulty in observing the bubbling process inside continuous casters. Bai and Thomas^[14] conducted water model experiments to visualize bubble formation during gas injection through a drilled hole into a vertical duct with downward, cross-flowing water, and measured the resulting initial bubble size for different air and water flow rates. They also developed a semi-analytical two-stage bubble-formation model to predict initial bubble size in argon-steel systems by inserting argon and steel properties. Yu and Zhu adopted this model to estimate initial bubble size for simulations of two-phase flow in a slab caster mold with and without the effects of an imposed magnetic field. The predicted flow patterns matched well with experimental measurements.^[9]

In this work, a two-step model system has been developed to simulate argon-steel twophase flows in a continuous caster. The first step is to determine the amount and location of gas entering the liquid steel and to estimate the initial bubble size distribution, based on a simulation of gas flow inside the UTN refractory. Using these results, the second step is to perform argonsteel two-phase simulations in the nozzle and mold regions of a conventional slab caster to obtain the two-phase flow patterns, comparing two multiphase flow models, an Eulerian-Eulerian model and an Eulerian-Lagrangian model, and different methods to compute the surface profile. Nail-board measurements of liquid steel surface velocities and mold level profiles were conducted to validate the computational models. The effects of initial bubble size and gas volume fraction on mold two-phase flow patterns were investigated. Finally, the models were applied to investigate the effect of different steel throughputs, gas volume fractions and mold widths on the flow pattern.

5.2 Plant Trials with Nail-board Experiments

Many methods and techniques have been proposed and applied to measure liquid steel velocities in continuous steel casters during operation.^[14-18] Among all these methods, the nail-dipping or nail-board method has proven to be a simple but powerful way of measuring both liquid steel surface velocities and mold level profiles in continuous casters, as found by previous work.^[17]

Plant measurements with nail-boards were conducted in this work to obtain the steel surface velocity and mold level across the mold width. During each experiment, the nail-board was held in a horizontal position, lowered vertically and dipped into the mold surface. Then the board was taken out of the liquid steel in a vertical path after about 3-5 seconds, which was needed for the liquid steel surrounding the each nail so solidify into a lump. To obtain the velocity distribution on the mold top surface, a correlation by Liu et al.^[19] was utilized in the current work to calculate the surface velocities from the measured nail diameters and height differences of the solidified steel lump around the nail perimeter, as given by Eq. (5.1).

$$V_s = 0.624 \phi_{lump}^{-0.696} h_{lump}^{0.567} \tag{5.1}$$

where V_s is steel surface velocity (m/s), ϕ_{lump} is lump diameter (mm), and h_{lump} is lump height difference (mm). This correlation to quantify liquid steel surface velocities was validated by another sets of measurement of surface steel velocities at the same location with a Sub-meniscus Velocity Control (SVC) device^[20] that was conducted simultaneously during a plant trial in ArcelorMittal Dofasco's No. 1 continuous caster.^[19] The nail-board approach has since been frequently used in ArcelorMittal to measure mold top surface steel velocities with satisfying performances.^[21] In addition to quantifying steel surface velocities, nail-boards also measure the mold top surface profiles and slag thicknesses, which are also important parameters for understanding mold flow phenomena and mechanisms for defects formation. In this study, mold top surface profiles are measured using nail-boards and are used to validate the computational models.

During the plant trials in Severstal, five sets of casting conditions were chosen for the nail-board experiments, with using a total of 16 nail boards. On average, 3-5 nail boards were dipped for each of the cases (casting conditions) listed. An example photo of a nail board after the experiment is shown in Figure 5.2. Three typical cases were chosen from the four scenarios for numerical simulations, with different casting speeds (steel throughputs), argon injection rates and mold widths, as given in Table I. Case 1 has a steel throughput of 3.0 tonne/min), wide mold width (1732 mm) and lowest argon injection rate, thus giving the least argon fraction (4%). Case 2 has the highest steel throughput (3.3 tonne/min) and argon injection rate, and lower mold width, giving 5.6% argon fraction. Case 3 has the same mold width as Case 2, but lowest casting speed (0.65 m/s), which leads to the lowest throughput (1.3 tonne/min) and highest argon volume fraction (12%). Three nail board experiments were conducted for each of these three cases, and in each nail board two rows of the nails totaling 20~26 nails were dipped, as shown in Figure 5.2. The lump height difference was measured on both the lump outer perimeter and around the nail. Thus two sets of surface velocity measurements were made for each dipped nail, which increases the number of samples for the statistics of surface velocity measurements to 12 measurements at each location.

5.3 Model Description and Computation Details

The two-step model system was used in the current work, which consists of a UTN gas porous-flow model to calculate hot argon flow rate and initial bubble size distributions exiting the UTN inner surface, and multi-phase flow models to simulate bulk argon-steel two-phase flow patterns in the nozzle and mold regions. The following two subsections explain each model.

5.3.1. Porous-flow Model of gas flow through heated UTN refractory

Uncertainties arise regarding the gas injection process due to practical difficulties in accurately measuring hot argon flow rate entering the liquid steel stream and multiple mechanisms that cause gas leakages. On the other hand, a quantitative understanding of this process is crucial to modeling the complicated argon-steel interactions in continuous casters and the resultant flow pattern. In order to gain the knowledge of gas flow through upper tundish nozzle refractory, mathematical models (e.g. a porous-flow model and a pressure-source model) were developed recently by Liu and Thomas^[22] to investigate argon distribution in the porous refractory subjected to the injection pressure, liquid metal hydrostatic pressure and refractory pore structure. This porous-flow model is adopted in the present work to study the gas flux through the refractory inner surface within a commercial UTN used in the Severstal conventional caster. Gas pressure distribution inside the porous refractory is governed by Eq. (5.2), with the effects of gas thermal expansion and temperature-dependent gas viscosity, in the laminar flow regime and subjected to a one-way coupled temperature field, *T*.

$$\nabla \cdot \left(K_D \nabla p \right) = -\frac{RT}{p} \left[\nabla \left(\frac{p}{RT} \right) \cdot \left(K_D \nabla p \right) \right]$$
(5.2)

where K_D is the permeability of the refractory material, p is the argon pressure in the refractory, R is the specific gas constant for argon. The gas velocity distribution is directly computed from the pressure (or pressure gradient) field following Darcy's law:

$$\mathbf{v}_a = -K_D \nabla p \tag{5.3}$$

It is worth pointing out that heat transfer is one-way coupled with the porous gas flow, in which the gas diffusion is assumed not to affect the heat transfer process. This assumption is supported by previous heat transfer calculation results by Bai and Thomas,^[6] which showed that gas flowing through the micro-channels in the refractory quickly heats up to the local temperature in the UTN. A one-way-flow pressure boundary condition given in Eq. (5.4) was applied at the UTN inner surface to during the solution procedure to prevent unphysical reverse gas flow into the refractory.

$$\begin{cases} if \quad \left(\frac{\partial p}{\partial n}\right)_{b} < 0, \qquad p_{b} = p_{l} + \sigma \kappa \\ else, \qquad \qquad \left(\frac{\partial p}{\partial n}\right)_{b} = 0 \end{cases}$$
(5.4)

where p is the gas pressure and varies along the interface boundary (subscript b). This one-way flow boundary condition in equation (s) seals portions of the boundary, which are unknown prior to the calculation. The choice of Dirichlet (first) or Neumann (second) boundary condition case in Eq. (5.4) is part of the model solution. Besides the thermal expansion effect accounted for in Eq. (5.2), gas viscosity varies with temperature, which in turn changes the permeability therefore the resistance to the flow. Gas at the refractory-steel interface must overcome the surface tension barrier to enter the liquid steel. This bubbling threshold is proportional to both the surface tension coefficient between the two phases and the size of the pore opening at the interface. This effect is also included in the current model. Further details of the porous-flow model are addressed elsewhere.^[22]

5.3.2. Predicting Initial Bubble Size Distribution

Initial bubble size, upon formation at the nozzle refractory/steel interface, is a key parameter to predict two-phase flow interactions in the nozzle and mold region in a continuous caster. Few studies have investigated this topic in depth due to practical difficulties in observing or measuring the bubbling process in molten metal. The semi-analytical two-stage model of bubble growth and detachment by Bai and Thomas^[14] to predict the initial bubble size is promising. However, in addition to the downward steel velocity, this model requires the gas velocity that exits each micro-hole or pore in the refractory wall. The superficial velocities of the argon gas over UTN inner surface calculated from the porous-flow model cannot be used directly, because the actual gas velocities that exit the micro-pores at the refractory surface are much larger. The many surface pore openings distributed on the inner surface of the UTN porous refractory where gas bubbles enter the liquid stream are termed "active sites" ^[23]. The number and locations of these active sites are needed to calculate the physical gas velocities that exit each pore and thereby link the porous-flow model and the two-stage bubble formation model. These parameters are difficult to determine, because the active sites depend on both refractory properties and flow conditions.

To measure active sites, Lee et al.^[24] conducted a series of water model experiments injecting air through a piece of refractory into downward flowing water in the nozzle, under different air and water flow rates. Before the experiment, a thin wax layer was deposited on the inner surface of the refractory sample to reproduce the non-wetting condition of a steelrefractory system. From the measurements, an empirical correlation was proposed to estimate the local number density of active sites:

$$N_{site} = \frac{7Q_{g}^{0.2635}U^{0.85}K_{DS}^{0.3308}}{\theta}$$
(5.5)

where Q_g is the gas injection flow rate (LPM/cm²), U is the mean liquid velocity (m/s), K_{DS} is the refractory specific permeability (nPm), and θ is the contact angle wettability (radians). N_{site} on the left side of the equation is the active sites number density (1/cm²).

In the current work, the porous-flow model for superficial gas velocity distribution[^{22]} is combined with the empirical correlation from Lee et al.^[24] for number density of active sites and the two-stage bubble formation model from Bai and Thomas, ^[23] for initial bubble size. The flow diagram in Figure 5.3 shows the procedure. First, the distribution of active sites on the UTN inner surface is calculated from the superficial gas velocity distribution, the steel velocity down the wall, and the refractory properties. Secondly, the physical gas flow rate through each active site, Q_{site} , is calculated from the pore size based on local mass conservation, following:

$$Q_{site} = \frac{Q_g}{N_{site}}$$
(5.6)

Finally, the two-stage bubble-formation model is used to predict the initial bubble size from the previous results.

5.3.3. Multiphase Flow Models

Mathematical models have been developed to describe the multiphase dynamics in gasliquid flow systems. The two-fluid models (e.g. Eulerian-Mixture and Eulerian-Eulerian) have been most frequently used to calculate velocity fields for multiple co-existing phases, in which the secondary (dispersed) phase is also treated as a continuous phase averaged in the same Eulerian framework. Eulerian-Lagrangian models have also been extensively used to simulate particulate and bubbly flows in dilute systems, with the dispersed phases (particles and/or bubbles) approximated as discrete point masses that interact with the continuous liquid phase. Models have also been developed using a front-tracking technique to resolve the bubble-liquid interface and the Eulerian model for the continuous liquid phase computation.^[25] However, this method can resolve only a small number (up to ~100) of bubbles so is prohibitive for a continuous caster which typically involves ~10⁴ bubbles. In this work, both Eulerian-Eulerian and Eulerian-Lagrangian models are evaluated to investigate the complicated two-phase flows in the Severstal continuous caster for the 3 cases in Table I.

5.3.3.1.Eulerian-Eulerian model

The two-fluid Eulerian-Eulerian model solves a set of continuity and momentum equations in each of the primary (liquid steel) and the secondary (argon gas) phases in the Eulerian frame of reference. Local gas concentrations are calculated based on the local number and volume of the gas bubbles in each computational cell and transport equations for gas momentum are solved accordingly. Continuity equations for both argon and liquid steel phases are given as follows.

$$\frac{\partial (\alpha_a \rho_a)}{\partial t} + \nabla \cdot (\alpha_a \rho_a \mathbf{v}_a) = 0$$
(5.7)

$$\frac{\partial (\alpha_s \rho_s)}{\partial t} + \nabla \cdot (\alpha_s \rho_s \mathbf{v}_s) = 0$$
(5.8)

Momentum equations for both phases are:

$$\frac{\partial (\alpha_a \rho_a \mathbf{v}_a)}{\partial t} + \nabla \cdot (\alpha_a \rho_a \mathbf{v}_a \mathbf{v}_a) = -\alpha_a \nabla p + \nabla \cdot (\alpha_a \mu_a \nabla \mathbf{v}_a) + K_{as} (\mathbf{v}_s - \mathbf{v}_a) + \alpha_a \rho_a \mathbf{g}$$
(5.9)

$$\frac{\partial (\alpha_s \rho_s \mathbf{v}_s)}{\partial t} + \nabla \cdot (\alpha_s \rho_s \mathbf{v}_s \mathbf{v}_s) = -\alpha_s \nabla p + \nabla \cdot (\alpha_s (\mu_s + \mu_t) \nabla \mathbf{v}_s) + K_{as} (\mathbf{v}_a - \mathbf{v}_s) + \alpha_s \rho_s \mathbf{g} \quad (5.10)$$

where the momentum transfer between the phases is achieved via the $K_{as}(\mathbf{v}_a - \mathbf{v}_s)$ source terms:

$$K_{as} = \frac{3}{4} \frac{C_D}{D_b} \alpha_s \rho_s \left| \mathbf{v}_s - \mathbf{v}_a \right|$$
(5.11)

Boundary conditions for the liquid steel phase consist of a fixed inlet velocity at the UTN top, a pressure outlet B.C. at the domain exit, and enhanced wall treatment^[26] on the no-slip walls, which include the nozzle side walls, shell interface and mold top surface. For the argon gas phase, a fixed velocity inlet B.C. is applied at the UTN inner surface and a "degassing" B.C.^[26] is used on mold top surface where the rising argon bubbles exit the domain. This degassing boundary condition applies mass and momentum sinks in the first layer of cells adjacent to the mold top surface, to account for the gas leaving the domain, and have been previously adopted in argon-steel two-phase mold flow simulations.^[10]

5.3.3.2.Eulerian-Lagrangian model

Momentum exchanges between the discrete bubbles and the liquid phase are calculated via two-way coupling, taking into account several different forces that act between the phases. A general formulation that describes spherical particle/bubble motion in the continuous liquid phase was proposed by Maxey and Riley,^[27] including the drag, lift, added-mass, pressure gradient, and gravity-related forces. These forces are calculated at each time step to obtain the instantaneous bubble acceleration and velocities. The particle transport equations solved in the current work are:

$$m_b \frac{d\mathbf{v}_b}{dt} = \mathbf{F}_D + \mathbf{F}_L + \mathbf{F}_{added-mass} + \mathbf{F}_G + \mathbf{F}_B + \mathbf{F}_{press-grad}$$
(5.12)

where \mathbf{v}_b is the bubble velocities, and m_b is the mass of the bubble. Forces on the right-hand side of Eq. (5.12) acting on the argon bubbles from the surrounding liquid steel include: drag (D), lift (L), added-mass, gravity (G), buoyancy (B), pressure and stress gradient forces.

Drag force is given by:

$$\mathbf{F}_{D} = C_{D} \frac{\pi \rho_{s} d_{b}^{2}}{8} |\mathbf{v}_{s} - \mathbf{v}_{b}| (\mathbf{v}_{s} - \mathbf{v}_{b})$$
(5.13)

where C_D is the drag coefficient, adopting the Schillar-Naumann^[28] formulation:

$$C_{D} = \frac{24}{Re_{b}} \left(1 + 0.15Re_{b}^{0.687} \right), \qquad Re_{b} = \frac{\rho_{s} \left| \mathbf{v}_{s} - \mathbf{v}_{b} \right| d_{b}}{\mu}$$
(5.14)

Lift force is calculated following according to Saffman,^[29]

$$\mathbf{F}_{L} = 1.61 d_{b}^{2} \left(\rho_{s} \mu \right)^{0.5} \left| \nabla \times \mathbf{v}_{s} \right|^{-0.5} \left[\left(\mathbf{v}_{s} - \mathbf{v}_{b} \right) \times \left(\nabla \times \mathbf{v}_{s} \right) \right]$$
(5.15)

Added-mass force is computed by:

$$\mathbf{F}_{added-mass} = C_A \rho_s V_b \left(\frac{D \mathbf{v}_s}{D t} - \frac{d \mathbf{v}_b}{d t} \right)$$
(5.16)

where C_A is the added-mass coefficient taking the value of 0.5 in the current work.

The gravity-related forces consist of the gravity force acting on the bubble and the buoyancy force exerted on the bubble by the surrounding liquid steel, which is given by Eq. (5.17).

$$\mathbf{F}_{G} + \mathbf{F}_{B} = \left(\rho_{b} - \rho_{s}\right) V_{b} \mathbf{g}$$
(5.17)

The pressure-gradient force is calculated via Eq. (5.18),

$$\mathbf{F}_{press-grad} = \rho_s V_b \frac{D \mathbf{v}_s}{D t}$$
(5.18)

It is worth mentioning that Eq. (5.18) refers to the dynamic part of the pressure gradient force, since the hydrostatic part of the pressure gradient force is already included in Eq. (5.17), as the buoyancy force, \mathbf{F}_{B} .

Two-way coupling between the discrete bubble tracking Eq. (5.12) and the momentum Eq. (5.10) of the continuous liquid steel phase (without gas volume fraction) is achieved by adding the following source term of the bubble reacting force into the momentum Eq. (5.10) for the liquid steel flows:

$$\mathbf{f}_{b} = -\frac{1}{V_{cell}} \sum_{i}^{N_{p,cell}} \left(\mathbf{F}_{drag} + \mathbf{F}_{lift} + \mathbf{F}_{press-grad} + \mathbf{F}_{added-mass} \right)_{i}$$
(5.19)

where *i* is the index for bubbles contained in the same control volume, and V_{cell} is the volume of the current control volume. Bubble-bubble interactions and bubble size evolution are neglected. The bubble size during the flow is assumed to stay constant and equal to its initial size upon formation in the UTN. Bubble velocity, \mathbf{v}_b , is integrated from Eq. (5.12), and the bubble position is then integrated from Eq. (5.20).

$$\frac{d\mathbf{x}_b}{dt} = \mathbf{v}_b \tag{5.20}$$

Boundary conditions for the discrete phase model include a "reflect" B.C. for bubbles bouncing at domain walls, and an "escape" B.C. for degassing at the mold top surface and domain outlet, where that bubble is no longer tracked.

5.3.4. Turbulence Models

Two different turbulence models are evaluated in the current work: 1) a shear stress transport (SST) k- ω model,^[30] and 2) a detached eddy simulation (DES) model,^[31] for the two different multiphase models, Eulerian-Eulerian and Eulerian-Lagrangian models, respectively. The SST k- ω model resembles the standard k- ε two-equation model for unsteady Reynolds averaged Navier-Stokes (U-RANS) simulation, but has different near-wall treatment with blending functions, providing a better performance predicting flows with adverse pressure

gradients and separations.^[30] The DES performs Large Eddy Simulation (LES) in the fluid bulk region to capture the local instantaneous turbulent eddies, and adopts a RANS wall function close to the no-slip walls to resolve the details of the turbulent boundary layer fluctuations with less computational effort than LES.

These two turbulence models are chosen to combine with different multiphase flow models: the k- ω model solves for an ensemble averaged velocity field, and the instantaneous small scale turbulent eddies are filtered out. So this URANS model is combined with the Eulerian-Eulerian model, where the averaged fields for both gas volume fraction and gas velocity are solved in the same reference frame with the continuous liquid steel. Therefore, the interactions between the argon bubbles with unresolved liquid steel eddies of similar sizes cannot be directly modeled in this model, since it is essentially difficult to extract instantaneous velocities from the Reynolds averaged flow field. Instead, models must be utilized to generate velocity fluctuations around the bubbles to estimate the local instantaneous velocities, such as the random walk model,^[26] which was adopted previously combining with a URANS model to track the dispersing particle inclusions in a continuous caster mold by Thomas et al.^[32] Since this model does not track individual gas bubbles, the size distribution of bubbles cannot be directly adopted in the model. Instead, a mean (effective) bubble size is used. This assumption is acceptable when liquid steel flow is not very sensitive to the bubble size. However, when the bubble size or its evolution during the flow significantly affects the resulting liquid steel flow patterns in the mold region, models should be adopted to account for the bubble size effect. On the other hand, the size distribution of injected argon bubbles can be naturally incorporated in the Eulerian-Lagrangian mdoel, since it resolves the velocity and trajectory of each bubble. Also, the interactions between the smaller scale eddies with the bubbles are directly computed with the
resolved instantaneous liquid steel velocities. Thus the Eulerian-Lagrangian multiphase model is combined with the DES model to reveal the details of this essentially transient process.

In this work, both turbulence models are applied to study the argon-steel flows with the two essentially distinct multiphase models.

5.3.5. Mold Level Tracking Models

Many methods have been used to predict mold top surface profiles,^[34-37] among which a simple pressure method is frequently used to calculate mold level.^[37] This method simply converts the local pressure into the liquid level elevation based on balancing potential energy, as given by Eq. (5.21), without actually changing the position of the free surface.

$$\Delta h = \frac{p - p_0}{\rho_s g} \tag{5.21}$$

where Δh is the mold level deviation, p is pressure along the top surface, and p_0 is the pressure at the reference mold level, which is taken at the mold quarter point from steady-state solution. ρ_s is the liquid steel density (7200 kg/m³), and g is gravitational acceleration (9.81m/s).

To more accurately simulate the top surface dynamics including gravity wave sloshing, and surface tension effects, a new interface tracking method is adopted from Liu et al^[39]. In this model, mesh nodes (or vertices) are moved to enforce the local mass conservation and satisfy the kinematic boundary condition at the interface.^[41]

$$\left[\left(\mathbf{v}-\mathbf{v}_{g}\right)\cdot\mathbf{n}\right]_{fs}=0$$
(5.22)

where **v** is the fluid velocity and **n** is the unit vector pointing to the surface normal direction. Subscripts *g* represents grid, and *fs* refers to interface between slag and steel. The grid is moved to satisfy Eq. (5.22) based on the local fluid velocities, using a method proposed by Peric.^[41] To maintain a good mesh, the interior nodes (vertices) are smoothed by solving for the local grid velocities as follows:^[26]

$$\nabla \cdot \left(\lambda \nabla \mathbf{v}_{g} \right) = 0 \tag{5.23}$$

where λ is a diffusion coefficient with the value 1.0 adopted in the current work. Further details are given elsewhere.^[26,39,41]

This method has the advantage over other approaches, such as volume of fluid (VOF) method^[38] of being easier to implement into a multiphase computational model such as the Eulerian-Eulerian model for argon-steel mold flow. In the current work, this model is compared with the simple pressure method to study free-surface shape in quasi-steady multiphase flow.

5.3.6. Computational Details

For this sophisticated model system, different numerical techniques are utilized for different sub-models, and the commercial package, ANSYS Fluent v15.0, is used for both the porous refractory and turbulent argon-steel flows models. For each of the three cases, the steady-state porous-flow model is first run to find the superficial argon gas velocity using the 3-D computational domain of the UTN in Figure 5.4, with a mesh of 0.6 million mapped hexahedral cells. Since a ring-slit is used to convey argon gas into the refractory, it is reasonable to simulate one quarter of the UTN based on the symmetry of the geometry, properties, and boundary conditions.

Next, a transient two-phase argon-steel simulation is conducted for each of the 3 quasisteady-state problems. For the Eulerian-Eulerian model, one half of the nozzle and mold was used as the computational domain, assuming left-right symmetry. The mesh, pictured in Figure 5.5, has 0.67 million mapped hexahedral computational cells, and a time step of 0.01 sec is adopted. The momentum equations (9) and (10) are discretized with a second order upwind scheme and numerical solutions are obtained with an implicit iterative algebraic multigrid solver.

For the Eulerian-Lagrangian model, the full nozzle/mold is chosen as the computational domain, as the DES model resolves instantaneous velocities for which symmetry does not apply. The PISO^[40] algorithm was used to solve the pressure Poisson equation. The momentum equations for DES simulations of liquid steel flows are discretized using a bounded central differencing scheme and are solved implicitly with the same techniques as used in the Eulerian-Eulerian model. A mesh of 1.8 million mapped hexahedral cells and a time step size of 0.005 sec were adopted. The steady solution was used as the initial condition. Argon bubbles were injected at each liquid steel flow step of each time step (0.01 sec). The bubble velocity is integrated from Eq. (5.12) using a trapezoidal rule, with a tracking time step of 1×10^{-3} sec.

For each multiphase simulation, a corresponding single phase flow simulation was performed to compare the flow patterns. Simulations were performed on the Blue Waters supercomputing clusters in the National Center for Supercomputing Applications (NCSA) in the University of Illinois at Urbana-Champaign (UIUC) to accelerate the computations. The UTN porous gas flow simulation was parallelized with 16 CPU processors (1 computing node), and the Eulerian-Eulerian multiphase simulations used 64 CPU processors (2 computing nodes). The full-mold Eulerian-Lagrangian parallel simulations used 16 computing nodes, consisting of 256 processors in total.

5.4 **Results and Discussion**

Simulation results for the 3 cases are presented in the following order: 1) gas velocity distributions through the porous UTN refractory, and the resulting bubble sizes; 2) single-phase

flow patterns using both the DES and the URANS models, comparing unsteady flow statistics between the two turbulence models; 3) two phase flow results in the nozzle and mold region, comparing the top surface velocities and profiles with the nail-board measurements. In addition, the two phase flow models are applied to study the effect of bubble size on the 3 flow patterns.

5.4.1. UTN Temperature, Gas Flow, and Initial Bubble Size

Temperature in the UTN refractory from the heat-transfer model is shown in Figure 5.6. Temperature increases from about 1000 K at the outer UTN surface to 1800 K on its inner surface. This temperature increase (800 K) across the ~30-mm-thick refractory wall causes the argon gas to expand as it flows through the UTN wall. Gas flow rates tend to be small and less important in the upper UTN, because it is far away from the ring-shaped distribution slit. Note, however, that the upper region of this UTN has a four-times larger inner bore diameter (120 mm), than the lower part (75 mm), which leads to four-times smaller downward cross-flow velocities in the upper region. The small cross-flow velocities tend to allow bubbles to grow larger in the upper UTN before they detach and may even cause large gas pockets to form, changing the flow regime from bubbly flow to slug flow. This might violate the assumption of downward bubbly flow, on which the bubble-formation is based, and lead to convergence problems. To resolve this issue, a smaller pore radius of 10 µm is assumed in the upper UTN (>0.05 m from UTN bottom), and a larger pore radius of 80 µm is chosen for the lower UTN region (<0.05 m). Inhomogeneous pore sizes are commonly seen in practice, due to compression variations during the UTN manufacturing process.

The calculated argon gas distributions for all three cases are shown in Figure 5.7 (a)-(c). High gas velocities are clearly seen in the lower regions of the refractory wall with zero-velocity

159

zones in the upper part of the UTN refractory. The small pore opening assumed in the upper UTN makes the local bubbling threshold too high for bubbles to form. Thus no pressure gradient exists within the upper refractory region, resulting in a no-flow region. All of the gas then enters liquid steel from the lower part of the UTN, where the bubbling threshold is lower due to the larger pore size. High gas velocity is predicted in all three cases near to the gas injection slit especially near the corner of the diameter change ~50 mm above UTN bottom. Case 2 has the highest gas velocities of the 3 cases, while Case 1 has the lowest velocities, and smallest injection rate.

Initial bubble size distributions (bubbling frequencies) are calculated with the procedure in Figure 3, and are shown in Figure 5.8 (a)-(c) for Cases 1-3 respectively. For these heat transfer and flow conditions, the bubble sizes typically range from 2 to 4.5 mm. The Case 1-2 distributions are predicted to have long tails on the large-bubble end, which is typical of a Roslin-Ramler particle-size distribution.

5.4.2. Single-phase Liquid Steel Flow Patterns

Flow pattern results from the single-phase DES simulation are presented in Figure 5.9, with (a), (c) and (e) showing instantaneous liquid steel velocity magnitude fields, and (b), (d) and (f) showing the time-averaged velocity fields in the mold center plane parallel to the broad faces. To obtain the mean velocities, statistics were collected for 40s, after waiting an initial ~25 sec for the quasi-steady flow field to develop.

A double-roll flow pattern is found in all the scenarios. Figure 5.9(a) and (b) show the instantaneous and time-averaged velocity distributions for Case 1. The chaotic nature of the turbulent liquid steels in the instantaneous velocity-contour snapshot shows significant left-right

asymmetry. The steel jet on the right side of the mold has a higher strength and bends upward with a stronger swirling zone leaving the SEN. The large-scale flow asymmetry is eliminated in the mean velocity distribution in Figure 5.9(b), by time-averaging over 40 sec.

Instantaneous velocity contours in Case 2 Figure 5.9(c) and Case 3 Figure 5.9(e) also show differences of flow fields between left and right regions of the mold. With a higher casting speed at the same mold width, steel throughput in Case 2 is about 2.5 times larger than in Case 3, which leads to more upward bending of the liquid steel jets. It also moves the centers of the upper recirculation regions towards the SEN.

For Case 1 with the largest mold width, the single-phase flow simulation results from the URANS model reveal an unstable jet that wobbles continuously, as shown in Figure 5.10(a)-(f). The flow pattern at 25 sec shows an upward-bending liquid steel jet that impinges on the mold top surface. The upper roll shows a strong recirculating region with the maximum surface velocity located close to the SEN. In the next frame, (28 sec), the liquid steel jet moves towards the mold upper corner and the surface velocity decreases. Next, (31 sec), the jet becomes wavy and impinges on the narrow face. Next, (36 sec), the path bends upwards, impinges on the narrow face, and the recirculating upper roll moves away from the SEN. Next, (42.5 sec), the jet bends further upwards and impinges near the mold upper corner, causing strong surface flow towards the SEN. Finally, (46 sec), the jet wobbles back towards the center of the top surface, with the maximum surface velocity returning close to the SEN. This completes a 20-sec jet oscillation cycle. This large scale jet instability with periodic oscillations and corresponding movements of the upper roll has been previously found in relatively large mold widths.^[42]

5.4.3. Argon and Liquid Steel Flow Patterns

5.4.3.1. Case 1

For the same steel throughput and mold width, the flow pattern for Case 1 in Figure 5.11 shows that gas injection (4% gas) lessens the extent of liquid steel jet wobbling, compared with the single-phase wobbling flow in Figure 5.10. The multiphase jet impinges continuously onto the narrow face and a more stable upper recirculation region is formed. Contours of gas volume fraction are shown in Figure 5.11 (b), with the maximum void fraction located near the SEN port exit. Rising gas bubbles near the SEN drag the flow upward and diffuse the liquid steel jet.

The measured and predicted steel surface velocity profiles are compared in Figure 5.12. Scattered symbols represent the measured surface steel velocity in the horizontal direction (parallel to the broad faces of the mold), averaged over all 12 samples. The simulated velocity profiles from the Eulerian-Eulerian model match reasonably well with the measurements in general, but the maximum horizontal steel velocities are underpredicted by ~20%. Figure 5.13 compares the measured and predicted mold level profiles, for both single phase and multiphase flows. Both measurements and simulation results suggest a depression of mold top surface around the mold quarter point. However, the peak-to-valley distance from the measurements is about 4 times higher than that of the prediction.

The mismatches in both top-surface velocities and mold level profiles are possibly explained by the remaining higher surface velocities during the decrease of casting speed, that was occurring when the measurements were taken. Alternatively, the surface level would match well with the predictions if the profile is rotated to correct a possible alignment error during the measurement.

5.4.3.2. Case 2

In Case 2, a narrower mold width and higher casting speed was used. Two different mean bubble sizes are considered (3-mm and 5-mm bubble diameters) in the Eulerian-Eulerian RANS simulations. A typical "double-roll" flow pattern is still seen for both bubble sizes (Figure 5.14). Gas velocities and volume fraction contours are shown in Figure 5.15(a) and (b). The smaller bubble size leads to a deeper and further penetration of the gas into the liquid steel pool. Also, as a feedback to the continuous phase, the smaller-sized bubbles (3mm) make the liquid steel jet less upward-bending, causing a lower jet impingement point on the narrow face compared to the 5mm bubble sizes, which further leads to a slightly lower surface velocity, as shown in Figure 5.14.

Top surface shapes with and without argon injection are compared using the moving-grid surface tracking algorithm in Figure 5.16. Without gas, Figure 5.16(a) shows that the maximum difference in level profile across the mold top surface is ~10mm. With 4% gas (5 mm bubbles), however, this maximum level difference reaches as high as 20 mm, due to increased surface velocity and gas rising near the SEN, as shown in Figure 5.16(b).

The surface velocity profiles calculated with both 3mm and 5mm bubble sizes are compared with nail-board measurements in Figure 5.17. The measured surface velocities agree well with all three calculations. However, some discrepancy is found in the velocity distribution, where measurements reach a maximum velocity close to the mold narrow face, while the calculated surface velocities are largest near the mold quarter point. Decreasing from 5mm to 3mm bubbles causes the surface velocity peak to move towards the narrow face, and matches closer to the measured velocity profile.

The measured top surface velocities in both inner and outer radius show a local minimum velocity about 0.25m from the mold/nozzle center. However, this "valley" does not appear in the

163

simulation results. Two sets of simulations were performed to evaluate the two different methods to predict the mold level profiles: one the simple pressure method, and the other a novel and accurate moving-grid free-surface tracking algorithm that is introduced in Chapter 6. Figure 5.18 compares the measurements with the calculated mold levels. Results using the two methods to predict top surface level profile match well with each other. This demonstrates that the simple pressure method is reasonable when the level variations are small enough not to affect the flow pattern. Surface profiles with both 3mm and 5mm bubbles also are close. The simulation with the 3 mm bubble diameter shows a shallower depression. Although the simulated mold levels show the same trend as the measurements, they have smaller elevations near the narrow face by ~10 mm. This discrepancy between measurements and calculation results might be explained by the same causes as for Case 1.

5.4.3.3. Case 3

Figure 5.19 shows the simulation results of steel and gas velocity distributions at the mold central plane for Case 3, which has the lowest steel throughput and highest argon gas volume fraction (12%). These conditions produce a complex flow pattern with reversed surface velocity (flow is directed to narrow face) on part of the mold top surface, due to the plume of gas bubbles rising near SEN, as shown in Figure 5.19(b). The mean flow pattern, Figure 5.19(a), consists of two recirculation regions above the jet: a major vortex from the impingement and deflection of the liquid steel jet on mold narrow face, and a small recirculation region near the SEN, caused by the rising argon bubbles.

The liquid steel velocity distribution on the mold top surface is shown in Figure 5.20(a), where the two regions of surface flows in opposite directions are clearly shown. These two

surface flows meet near the center of the top surface. Figure 5.20(b) shows clearly that the region of reverse flow near the SEN is where the gas exits the domain.

Simulations with three different bubble sizes (3mm, 5mm and 8mm), produce horizontal surface velocity profiles that agree closely with each other, and all match very well with measurements, as shown in Figure 5.21. This indicates that the surface velocities for these casting conditions are not very sensitive to the bubble size. The transition between the positive surface horizontal velocities (directed from narrow face to SEN) and negative velocities (reverse flow towards the narrow face) is also well predicted. The magnitude of the positive surface velocities is significantly decreased from Case 2, due to the drop in steel throughput. A comparison of mold level profile between simulation results and measurements is given in Figure 5.22, using only the simple pressure method. All of the results match well with each other in general. However, a mismatch occurs near the SEN, where the measurements show an increase of the mold level (above the average mold level) due to the rising bubble plume, which is not captured by the simple pressure method, which shows a drop.

5.5. Conclusions

A model system has been established in this work to simulate argon-steel two phase flows in a continuous steel caster, consisting of: 1) a porous-flow model to simulate gas distribution in the UTN porous refractory, 2) a prediction of the argon flow rate and initial bubble size distribution based on the flow conditions, and 3) multiphase computational models to study argon and steel flow patterns in a caster mold region. Plant trials were conducted to measure top surface steel velocities and level profiles using the nail board approach, which serves as a powerful tool to understand flow behavior in a caster mold and also validates computational models. Based on the simulations and nail board plant experiments, the following conclusions can be drawn:

- 1. The porous-flow model together with the bubble size calculation gives quantitative insight into the gas injection process and provides reasonable estimates of two key parameters, argon flow rate in hot condition and initial bubble size (distributions), in understanding the argon-steel two phase flow phenomena.
- 2. Argon injection has a huge effect on the liquid steel flow pattern in the mold region. For double-roll flow patterns, increasing (bulk) argon volume fraction decreases mold top surface steel velocities towards SEN, and even causes reverse flow (towards to the narrow face) on part of the mold top surface. This is consistent with previous findings.^[19] The current model system successfully predicts this flow transition.
- 3. With typical gas volume fractions (4-5%), smaller bubbles (thus more bubbles for a given gas fraction) tend to lower surface velocities (by ~10%); while with high gas volume fractions (>10%), bubble size within the range of this study (3-8 mm) did not have much influence on the flow pattern.
- 4. Both the simple pressure method and the moving-grid free surface tracking algorithm for mold level calculations are validated by the nail board measurements. The simple pressure method is shown to be quite reasonable for the small-amplitude, quasi-steady state level variations in this typical caster.

5.6. Tables and Figures

Case #	Casting Speed (m/min)	Mold Width (mm)	Mold Thickness (mm)	Argon Flow Rate (SLPM)	Back Pressure (psi)	Submergence Depth (mm)
1	1.15	1732		5.02	15.45	
2	1.65	1397	221	7.0	18.07	222
3	0.65	1397		6.3	19.18	

Table I. Casting Conditions for Nail Board Trials

Table II. UTN Refractory Properties

Tundish level	Surface Tension	Mean PoreRadius	Permeability
(m)	σ (N/m)	(µm)	(npm)
0.70	1.157	50	10.1



Figure 5.1. Schematic of transport phenomena in continuous casting mold



Figure 5.2. Photo of a dipped nail board



Figure 5.3. Flow chart for initial bubble size prediction procedure



Figure 5.4. Schematic of UTN domain and mesh



Figure 5.5. Domain and mesh for mold flow simulations



Figure 5.6. UTN temperature field



Figure 5.7. Gas velocity distribution in UTN refractory



Figure 5.8. Initial bubble size distribution



Figure 5.9. Single phase steel flow pattern at mold center plane (DES results)



Figure 5.10. Single phase flow results (URANS model)



Figure 5.11. Two-phase flow results – mold centerplane (Case 1)



Figure 5.12. Mold top surface velocity profiles – Case 1













Figure 5.17. Top surface steel velocity profiles - Case 2



Figure 5.18. Mold top level profile – Case 2



Figure 5.19. Center plane velocity distribution – Case 3



Figure 5.20. Top surface steel velocity Distribution – Case 14



Figure 5.21. Top surface velocity profiles – Case 3



Figure 5.22. Mold level profile – Case 3

5.7. References

- B.G. Thomas, "Chapter 5. Modeling of Continuous Casting", *Making, Shaping and Treating of Steel*, 11th Edition, Vol. 5, Casting Volume, A. Cramb, ed., The AISE Steel Foundation, Pittsburgh, PA, 2003, pp. 1-24.
- [2] L. Zhang, S. Yang, X. Wang, K. Cai, J. Li, X. Wan, and B.G. Thomas, "Investigation of Fluid Flow and Steel Cleanliness in the Continuous Casting Strand", *Metallurgical and Materials Transactions B*, 38B(1), 2007, pp. 63-83.
- [3] R. Sánchez-Perez, R. D. Morales, M. Díaz-Cruz, O. Olivares-Xometl and J. Palafox-Ramos, "A Physical Model for the Two-Phase Flow in a Continuous Caster Mold", *ISIJ International*, 43(5), 2003, pp. 637-646.
- [4] X. Huang and B.G. Thomas, "Modeling of Transient Flow Phenomena in Continuous Casting of Steel", *Canadian Metallurgical Quarterly*, 37(304), 1998, pp. 197-212.
- [5] B.G. Thomas, X. Huang and R. C. Sussman, "Simulation of Argon Gas Flow Effects in a Continuous Slab Caster," *Metallurgical and Materials Transactions B*, 25B(4), 1994, pp. 527-547.
- [6] H. Bai and B. G. Thomas, "Turbulent Flow of Liquid Steel and Argon Bubbles in Slide-Gate Tundish Nozzles, Part I, Model Development and Validation", *Metallurgical and Materials Transactions B*, 32B(2), 2001, pp. 253-267.
- [7] T. Shi and B.G. Thomas: "Effect of Gas Bubble Size on Fluid Flow in Continuous Casting Mold", Continuous Casting Consortium at University of Illinois at Urbana-Champaign, Report, 2001.

- [8] V. Singh, S.K. Dash, J.S. Sunitha: "Experimental Simulation and Mathematical Modeling of Air Bubble Movement in Slab Caster Mold", 2006, *ISIJ International*, vol. 46(2), pp. 210-218.
- [9] H. Yu and M. Zhu: "Numerical Simulation of the Effects of Electromagnetic Brake and Argon Gas Injection on the Three-dimensional Multiphase Flow and Heat Transfer in Slab Continuous Casting Mold", *ISIJ International*, 2008, 48(5), pp. 584–591.
- [10] R. Liu, B.G. Thomas and J. Sengupta, "Simulation of Transient Fluid Flow in Mold Region during Steel Continuous Casting," *Modeling of Casting, Welding, and Advanced Solidification Processes (MCWASP) XIII*, (Styria, Austria, June 17-22, 2012), 2012.
- [11] M. Burty, M. Larrecq, C. Pusse, and Y. Zbaczyniak: "Experimental and Theoretical Analysis of Gas and Metal Flows in Submerged Entry Nozzles in Continuous Casting", *Revue de Metallurgie. Cahiers D'Informations Techniques*, 1996, vol. 93(10), pp. 287-292.
- [12] Z. Liu, B. Li, M. Jiang and F. Tsukihashi, "Modeling of Transient Two-Phase Flow in a Continuous Casting Mold using Euler-Euler Large Eddy Simulation Scheme", *ISIJ International*, 53(3), 2013, pp. 484-492.
- [13] H. Bai and B.G. Thomas: "Bubble Formation during Horizontal Gas Injection into Downward Flowing Liquid", *Metallurgical and Materials Trans. B*, 2001, vol. 32B(6), pp. 1143-1159.
- [14] P.H. Dauby, W.H. Emling, and R. Sobolewski, "Lubrication in the Mold: A Multiple Variable System", *Ironmaker and Steelmaker*, 1986, 13, p. 28-36.
- [15] M. Iguchi, et al., "A New Probe for Directly Measuring Flow Velocity in a Continuous Casting Mold", *ISIJ International*, 1996, 36(Suppl.), pp. S190-S193.

- [16] R. McDavid and B.G. Thomas, "Flow and Thermal Behavior of the Top-Surface Flux/ Powder Layers in Continuous Casting Molds" *Metallurgical and Materials Transaction B*, 1996, 27B(4), pp. 672-685.
- [17] K. Cukierski and B.G. Thomas, "Flow Control with Local Electromagnetic Braking in Continuous Casting of Steel Slabs", *Metallurgical and Materials Transactions B*, 39B(1), pp. 94-107.
- [18] B. Rietow and B.G. Thomas, "Using Nail Board Experiments to Quantify Surface Velocity in the CC Mold", Proc. of AISTech 2008 Steelmaking Conf., Pittsburgh, PA, (2008).
- [19] R. Liu, J. Sengupta, D. Crosbie, S. Chung, M. Trinh and B.G. Thomas, "Measurement of Molten Steel Surface Velocity with SVC and Nail Dipping during Continuous Casting Process", *TMS Annual Meeting*, San Diego, CA, 2011, pp. 51-58.
- [20] J.F. Domgin, P. Gardin, J.M. Galpin and A. Dez, "Effect of Process Parameters Variation on CC Mold Hydrodynamics and Inclusion Behavior", *Revue de Metallurgie-CIT*, 10, 2005, pp. 703-710.
- [21] J. Sengupta et al., "Direct Measurement of Sub-meniscus Velocity at ArcelorMittal Dofasco' s No. 1 Continuous Caster to Evaluate Mold Fluid Flow at Different Casting Conditions", *Proc. of AISTech Conf.*, Indianapolis, IN, 2011, pp. 1697-1706.
- [22] R. Liu and B.G. Thomas: "Model of Gas Flow through Upper Tundish Nozzle Refractory and Initial Bubble Size", *Proc. of AISTech 2012 Steelmaking Conf.*, Atlanta, GA, 2012, pp. 2235-2246.
- [23] G. Houghton, A.M. McLean, and P.D. Ritchie: "Mechanism of formation of gas bubblebeds", *Chemical Engineering Science*, 1957, 7(1–2), pp. 40-50.

- [24] G. Lee, B.G. Thomas and S. Kim, "Effect of Refractory Properties on Initial Bubble Formation in Continuous-casting Nozzles", *Metal and Materials International*, 2010, vol. 16(3), pp. 501-506.
- [25] G. Tryggavason et al., "A Front-Tracking Method for the Computations of Multiphase Flow", *Journal of Computational Physics*, 169, 2001, pp. 708-759.
- [26] FLUENT ANSYS Inc. v14.4-Manual (Lebanon, NH).
- [27] M.R. Maxey and J.J. Riley, "Equation of Motion of a Small Rigid Sphere in a Nonuniform Flow", *Physics of Fluids*, 26, 1983, pp. 883-889.
- [28] L. Schiller and Z. Naumann, "A Drag Coefficient Correlation", Z. Ver. Deutsch. Ing., 77, 1935, pp. 318-320.
- [29] C. Crowe, M. Sommerfild, and Y. Tsinji, "Multiphase Flows with Droplets and Particles", CRC Press, Boca Raton, FL, pp. 23-95.
- [30] F.R. Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", *AIAA Journal*, 1994, 32(8), pp 1598.
- [31] P.R. Spalart, W.H. Jou, M. Strelets and S.R. Allmaras, "Comments on the Feasibility of LES for Wings and on the Hybrid RANS/LES Approach", Proc. of the 1st AFOSR Int. Conf. on DNS/LES, Arlington, TX, 1997.
- [32] B.G. Thomas, Q. Yuan, S. Mahmood, R. Liu and R. Chaudhary, "Transport and Entrapment of Particles in Steel Continuous Casting", *Metallurgical and Materials Transactions B*, 2014, 45(1), pp. 22-35.
- [33] E. Krepper, T. Frank, D. Lucas, H.M. Prasser and Ph.J. Zwart, "Inhomogeneous MUSIG
 Model A Population Balance Approach for Polydispersed Bubbly Flows", *the 12th*

International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-12), Pittsburgh, Pennsylvania, 2007.

- [34] Y.H. Ho, C.H. Chen and W.S. Hwang, "Analysis of Molten Steel Flow in Slab Continuous Caster Mold", *ISIJ International*, 34, 1994, pp. 255-264.
- [35] G.A. Panaras, A. Theodorakakos and G. Bergeles, "Numerical Investigation of the Free Surface in a Continuous Steel Casting Mold Model", *Metallurgical and Materials Transactions B*, 29B(5), 1998, pp. 1117-116.
- [36] J. Anagnostopoulos and G.Bergeles, "Three-Dimensional Modeling of the Flow and the Interface Surface in a Continuous Casting Mold Model", *Metallurgical and Materials Transactions B*, 30B(6), 1999, pp. 1095-1105.
- [37] Q. Yuan, B.G. Thomas and S. P. Vanka, "Study of Transient Flow and Particle Transport during Continuous Casting of Steel Slabs, Part 1. Fluid Flow", *Metallurgical and Materials Transactions B*, 35B(4), 2004, pp. 685-702.
- [38] C.W. Hirt and B.D. Nichols, "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries", *Journal of Computational Physics*, 39, 1981, pp. 201-225.
- [39] R. Liu, B.G. Thomas, L. Kalra, T. Bhattacharya and A. Dasgupta, "Slidegate Dithering Effect on Transient Flow and Mold Level Fluctuations", *Proc. of AISTech Conf.*, 2013, pp. 1351-1364.
- [40] R.I. Issa, "Solution of the Implicitly Discretised Fluid Flow Equations by Operator-Splitting", *Journal of Computational Physics*, 62, 1986, pp. 40-65.
- [41] S. Muzaferija and M. Peric, "Computation of Free-Surface Flows using the Finite-Volume Method and Moving Grids", *Numerical Heat Transfer, Part B: Fundamentals*, 1997, 32(4), pp. 369-384.

[42] R. Liu, W. Ji, J. Li, H.F. Shen and B.C. Liu, "Numerical Simulation on Transient Flow Patterns of Upper Rolls in Continuous Slab Casting Moulds", *Steel Research International*, 79, 2008, pp. 50-55.

CHAPTER 6.

MODELING TRANSIENT MULTIPHASE FLOW AND TOP SURFACE DYNAMICS IN CONTINUOUS CASTER MOLD DURING SLIDE-GATE DITHERING

6.1. Introduction

The quality of the final product in continuous steel casting is closely related to the complicated transport phenomena coupling with turbulent liquid steel flows in the mold region.^[1] Large scale transient events during continuous casting are often detrimental to product quality.^[2] Defects are more likely to form during sudden changes of flow pattern in the nozzle and mold region. caused by essentially transient events such as a casting start-up,^[3] casting speed increase or decrease,^[4] release of clogged material from the SEN into the liquid steel stream^[5] and sudden movements of the actuator, i.e. stopper-rod^[6] or slide-gate.^[7] Thus, there is a great need to investigate these transient processes, to gain insight into the behavior of transient turbulent multiphase flows in the nozzle/mold region during continuous casting.

As large-scale flow variations in the nozzle and mold region are induced by sudden changes of liquid steel/argon flow rate in the system, the first step to study essentially-transient processes is to quantify these flow rate variations for both phases. Due to practical difficulties in measuring liquid steel flow rate in continuous casters, mathematical models have been developed in previous work to predict this flow rate, based mainly on the actuator (stopper-rod and slide-gate) position history.^[6-8] Argon gas is usually injected into conventional continuous casters to prevent clogging and re-oxidation of the liquid steel by entrained ambient air.^[9] The argon also interacts with liquid steel in the nozzle and mold region and can significantly change the liquid steel flow pattern, especially with high argon volume fractions. Argon injection rate is usually

measured in the "cold" condition (1 atm pressure and room temperature) far away from the UTN. This rate is very different from that entering the liquid steel stream, for two reasons: 1) gas expands while diffusing through the porous refractory wall of the UTN with temperature increase; 2) gas leakage occurs in the delivery system, such as through the joints between different refractory pieces or through possible cracks in the refractory. Thus, neither the liquid steel nor the argon gas flow rate during these transient events can be accurately measured.

Owing to the difficulty in measuring steel and argon flow during the real continuous casting process, water model experiments have been used extensively to study the air-water two-phase flow patterns in the mold region.^[10-12] However, as previously found by Huang and Thomas,^[5] results from water model experiments can be very different from the real casting process, due to differences in the material properties, such as gas viscosity, surface tension, contact angles, resulting bubble size distributions, and in the heat transfer conditions, the solidifying shell, and the mold bottom. Thus studies in cold laboratory models alone are not sufficient to fully understand the physics in the real caster, especially under multiphase flow conditions.

Mathematical modeling has become a necessary tool to study multiphase flow in continuous casting, including the prediction of argon and steel flow rate variations to provide reasonable inlet boundary conditions for subsequent flow simulations of the nozzle and mold region. Transient flow in this system has been investigated in a few modeling efforts, using 3-D Unsteady Reynolds Averaged Navier-Stokes (URANS) models and recently Large Eddy Simulations.^[13-18] However, most of the previous transient simulations focused on modeling quasi-steady state flows,^[10-18] and only a few have studied essentially transient processes.^[3-7] Wang and Zhang^[3,4] performed URANS simulations to study transient flow without argon during

a casting startup,^[3] and during a speed change^[4] using a volume of fluid (VOF)^[19] method to track the mold top surface.

One common cause of flow rate variations in continuous casting is clogging in the nozzle. Nozzle clogging limits production, can send large inclusions into the mold when clogs dislodge,^[6] and is detrimental to the flow pattern. Clogging can increase detrimental meniscus level fluctuations,^[20] and cause defects in the final product, including surface defects, slag entrainment via many different mechanisms, and internal inclusions. Huang and Thomas^[5] developed a 3-D finite-difference model to simulate transient argon-steel two phase flow patterns in the mold, and identified the phenomenon of large scale vortex shedding during the transition from asymmetric flow with nozzle clogging to steady symmetrical flow after the clog released. Sudden release of a sticking slide gate after periods of inactivity is another cause of detrimental flow variations.

In order to avoid these issues, "dithering" of the slide gate has been implemented in some operations by oscillating the middle sliding plate back and forth with a small stroke and high frequency.^[20] This continuous motion prevents sticking and may also reduce clogging. However, it also causes flow-rate variations.^[21] So slide-gate dithering has a direct impact on mold level fluctuations, which is crucial to defect formation and product quality. Thus, it is of great importance to investigate the effect of the slide-gate dithering on the transient bulk flow in the nozzle and mold, to quantify the extent of top surface level variations, and to gain insights to further understand and improve this method.

Previous work on dithering has focused mainly on measurement of mold level.^[20-21] Control systems must be altered to maintain constant average flow rate and stable mold level, and special control algorithms have been developed to accomplish this.^[21] Unfortunately, the time-variation of flow rate cannot be measured in plant trials, even though it is a critical parameter in mathematical modeling to investigate phenomena during realistic transient flow events. A modeling approach to determine the flow rate history based on the stopper-rod position history was recently developed by Liu et al.^[6] and applied to study the effects of stopper rod movements on flow variations that eventually lead to sliver formations at ArcelorMittal Dofasco. In the current work, the same approach was used to develop a new model to predict the steel flow rate, based on the recorded slide-gate position history, as well as other process parameters, such as the nozzle/plate diameter, tundish level, and submergence depth.

To better quantify the UTN argon injection rate and the resulting initial bubble sizes, Liu and Thomas^[23] recently developed a porous-flow model to calculate argon pressure and velocity distributions through the UTN refractory and at its inner surface. This gas exits certain "active sites" or pores in the inner surface of the UTN refractory to form bubbles.^[24] Lee et al.^[25] found an empirical correlation to estimate the number density distribution of active sites on a refractory surface for different water flow rates, refractory properties and gas injection rates. This correlation has been combined with a two-stage model by Bai and Thomas^[26] to predict the initial bubble size, based on the gas velocity distributions computed by the porous-flow model.^[21]

Upon obtaining these gas flow parameters, argon-steel two-phase flows are simulated using an Eulerian-Eulerian model solving for two sets of the gas and liquid momentum equations in the mold bulk region. In order to simulate the dynamic response of the mold top surface, a free-surface tracking algorithm such as the well-known volume of fluid (VOF) method is required to couple with bulk flow solutions. However, the VOF method is difficult to implement together with a separate multiphase flow model for the argon gas bubbles in the molten steel. Thus a new free-surface tracking method was developed using a moving-grid approach, which is compatible with any finite-volume-based model of multiphase flow in the bulk region.

This work presents a system of models as discussed above to investigate transient flow behavior during continuous casting, and applies it to study slide-gate dithering effects for both single- and multi- phase flows. First, the flow-rate history is calculated via a gate-position-based model, which is validated with both water model experiments and plant measurements. Next, the argon volumetric flow rate in hot condition is calculated by a porous-flow model of the gas diffusing through a heated refractory, and further used to predict the velocity and bubble-size distributions entering from the UTN inner surface. Then the 3-D transient models, that were validated in Chapter 5, are applied to gain new insight into both single-phase and argon-steel two-phase flows in the SEN and mold region during the dithering process, based on inlet conditions from the previous models. The new free-surface tracking algorithm is then implemented into the argon-steel two phase Eulerian-Eulerian flow models and applied to further investigate the mold flow pattern evolutions and top surface motion under slide-gate dithering and argon injection. The model predictions are compared with plant measurements obtained during slide-gate dithering trials.

With this well-validated model system, a total of four simulations are performed in the current work to investigate slide-gate dithering, including single-phase flows at three different dithering frequencies, and two-phase argon-steel flows at two different dithering frequencies. Simulated mold level fluctuations are compared with measurements using an eddy current sensor. Combining the modeling results and plant experiments reveals new insight into the mold flow and top surface variations during the slide-gate dithering process.

188

6.2. Plant Measurements

Full-scale commercial-production trials were conducted at ArcelorMittal Indiana Harbor 3SP, to investigate various slide-gate dithering conditions (dithering frequency and stroke) for different casting speeds and mold widths both with and without argon injection. The SEN had an 80 mm inner bore diameter, 140 mm outer diameter, a roof-type bottom, 80 mm diameter round ports with a downward angle of 15 deg., and a 75 mm slide-gate plate opening diameter. An eddy-current sensor recorded the mold level history near the mold quarter point. Numerical simulations are performed in the current work for four different cases, shown in Table I.

An example of the results is given in Figure 6.1, showing the histories of key variables gate position, mold level, and dither frequency during trials with only 1% argon injection. Large sloshing waves were observed in an 1840 mm (72.5 inch) width mold at a dithering frequency of 0.9 Hz, which prompted operators to switch to manual level control for safety. The gravity wave that caused this large amplitude sloshing in the mold could not be predicted in previous models.^[27]

The effect of dithering frequency and stroke on these mold level fluctuations is further plotted in Figure 6.2. Both the maximum and the root mean square (*rms*) variation of the measured mold level fluctuations are shown. For most frequencies (other than 0.9 Hz), the mold level fluctuations are affected only by the dithering stroke, and dithering frequency has little effect. However, when the dithering frequency was 0.9 Hz, (1840 mm width), both the *rms* and maximum mold level fluctuations jumped to about 4-5 times larger than those from all the other cases. These giant mold level fluctuations (measuring over 10 mm from peak to valley) were identified to be caused by severe mold sloshing waves. One of the purposes of this work is to create and apply a system of computational models to investigate the effect of dithering on mold

level fluctuations, including the prediction and understanding of the sloshing mechanism that causes the giant level fluctuations.

A preliminary analysis of this excited mold sloshing shows that for this mold width and mold thickness, a natural frequency could be calculated to roughly match with the dithering frequency of 0.9 Hz, via Eq. (6.1). Sloshing in the mold resembles that in a rectangular tank, which has been well investigated by previous researchers.^[28-29] The natural frequency for the tank sloshing problem is given by equation (1) below, and pictured in Figure 6.3.

$$f_{i,j}^{2} = \frac{g}{4\pi} \sqrt{\left(\frac{i}{a}\right)^{2} + \left(\frac{j}{b}\right)^{2}} \tanh\left[\pi h \sqrt{\left(\frac{i}{a}\right)^{2} + \left(\frac{j}{b}\right)^{2}}\right]$$
(6.1)

where f (Hz) is the natural frequency, i and j are sloshing mode indices along the x and y-axis respectively, a, b and h, (m) are the tank dimensions in the x, y and z-directions respectively, g is gravity acceleration (9.81 m/s²).

The sloshing frequencies for different mold dimensions using Eq. (6.1) are plotted in Figure 6.4. Because the caster is deep, the *tanh* term in Eq. (6.1) is about 1, so can be neglected. For a slab width of 1840 mm, (72.5 inch), the first-mode, half-width from SEN to NF (0,1), or the second-mode, full-width from NF to NF (0,2), the sloshing frequency from Figure 6.4 is 0.92 Hz in both cases, which is very close to the sloshing frequency observed in the plant. Thus, Figure 6.4 shows the mold width / dithering frequency combinations to avoid. Other specific frequencies to avoid are 1.17 Hz for a 1140-mm (45-inch) wide slab, 1.1 Hz for a 1270-mm (50-inch) wide slab, and 1.0 Hz for a 1550-mm (61-inch) wide slab.

Plant trials were also conducted to investigate the combined effects of argon injection and slide-gate dithering on the mold flow and top surface variations. During slide-gate dithering, the liquid steel pressure varies inside the nozzle, which affects the gas velocity and related bubble-

size distributions. Varying these parameters increases the difficulty from a modeling perspective, and separate models were developed to study this behavior.

Table I shows the casting conditions of the four dithering cases chosen for numerical studies in the present work. Case 1-2 are chosen without gas injections for two different dithering frequencies: 0.9 and 1.4 Hz respectively. In order to avoid any potential control issues, the dithering stroke was set to 7 mm for the high dithering frequency Case 3 of 1.4 Hz, while remained 14 mm for the other dithering frequencies. Case 4 and 5 adopt a low dithering frequency of 0.4 Hz, but with different dithering strokes of 14 (Case 4) and 7 (Case 5) mm with argon injections.

6.3. Model Description

A system of models has been developed to simulate the complex phenomena associated with essentially transient, turbulent, multiphase flow in the nozzle and mold and the resulting mold level fluctuations, and is applied to investigate the effects of slide-gate dithering, as shown in Figure 6.5. First, liquid steel flow rate through SEN is predicted based on the slide-gate positions recorded during dithering using an analytical flow rate model, developed in the current work. Next, the porous-gas flow model is used to quantify the hot argon flow rate and velocity exiting the UTN inner surface, and the corresponding bubble size distribution. Then, the flow pattern in the SEN and mold are computed by solving the transient 3-D Navier-Stokes equations using the shear stress transport (SST) k- ω model for turbulence.^[30,43] To calculate the surface behavior properly, including level fluctuations, sloshing and the effects of surface tension, the moving shape of the interface needs to be calculated accurately, and without introducing extra phases into the multiphase model, a new free-surface tracking algorithm with a moving-grid
technique is developed and integrated into the commercial CFD package of ANSYS Fluent (v14.5) combining its dynamic mesh feature.^[30] This model has been validated and implemented to simulate the motion and deformation of the mold top surface.

In addition to constructing the 3-D transient (multiphase) CFD model, a simple analytical model was also developed based on liquid steel mass conservation, to estimate the change of the average mold level during the dithering process. All of these models were validated with both experimental measurements and analytical solutions, and then applied to investigate the transient flow pattern evolution and mold level fluctuations during slide-gate dithering.

6.3.1. Gate-position-based Flow Rate Model

The liquid steel flow rate through the upper tundish nozzle (UTN) and SEN varies with slide-gate position during dithering. An analytical model, Eq. (6.2), has been derived from Bernoulli's equation to predict the volumetric flow rate based on casting parameters such as tundish height, SEN inner bore diameter, port area, length of SEN, in addition to the slide-gate position. The parameter C is the clogging factor, which is set to 0 in the current study (assuming no clogging).

$$Q_{SEN} = A_{eff} \sqrt{\frac{2g(H_1 + H_2)}{\left(\frac{A_{SEN}}{A_{port}} - 1\right)^2 + f\frac{L_{SEN}}{D_{SEN}} + \left(\frac{1}{\mu} - 1\right)^2 \left(\frac{A_{SEN}}{A_{GAP}}\right)^2 + \left(\frac{A_{SG}}{A_{GAP}} - \frac{A_{GAP}}{A_{SG}}\right)^2 \left(\frac{A_{SEN}}{A_{SG}}\right)^2 + \left(\frac{A_{SEN}}{2A_{port}}\right)^2 + C}$$
(6.2)

where Q_{SEN} (m³/s) is the calculated time-dependent fluid volumetric fluid flow rate in UTN / SEN, H_1 (m) is the distance between the tundish level and bottom of the tundish, H_2 (m) is the distance from UTN upper edge to the upper edge of SEN port exit, g is the gravitational acceleration (m/s²), A_{SEN} (m²) is the SEN inner cross-section area, A_{port} (m²) is the area of the port exit projected to the liquid steel jet direction, f is the friction factor for turbulent flow in a circular pipe, with an estimated value of 0.075 from the Moody's chart, L_{SEN} (m) is the SEN length, D_{SEN} (m) is the SEN inner diameter, (90mm in water model; 80mm in caster), μ is the coefficient of contraction (venu contracta), A_{SG} (m²) is the area of the slide gate opening when it is fully opened, A_{GAP} (m²) is gap opening area projected in the casting direction, which depends on measured slide gate position.

The gap area is calculated from the recorded time-dependent gate opening distance, D, and the two diameters D_1 and D_2 , as follows:

$$A_{GAP} = \frac{D_1^2}{4} \arcsin\left(\frac{2h}{D_1}\right) + \frac{D_2^2}{4} \arcsin\left(\frac{2h}{D_2}\right) - Dh, \qquad if \quad D > \frac{\sqrt{D_1^2 - D_2^2}}{2}$$
(6.3)

where *h* is given by:

$$h = \frac{D_1 D_2}{4D} \sqrt{1 - \left(\frac{D_1^2 + D_2^2 - 4D^2}{2D_1 D_2}\right)^2}$$
(6.4)

The two diameters D_1 and D_2 are defined such that $D_1>D_2$, and represent the plate bore diameter and SEN inner diameter, depending on which is larger. As shown in Figure 6.6, when the two circle centers approach each other during gate opening, D decreases, the overlapped area A_{GAP} becomes larger, and flow rate increases. If the two diameters, D_1 and D_2 , are very different, then for large nozzle opening fractions, D may become small enough that the condition in Eq. (6.3) does not hold, and the gap area can no longer be calculated using this equation. However, this scenario is not expected during normal casting operations, so only Eq. (3) is presented.

The parameter μ in Eq. (6.2) is the ratio between the area of the Vena contracta and A_{GAP} , and is estimated with Eq. (6.5) below,^[31] Note that μ here is not the dynamic viscosity for fluids:

$$\mu = 0.63 + 0.37 \left(\frac{A_{GAP}}{A_{SG}}\right)^3$$
(6.5)

 A_{eff} in Eq. (6.2) represents the effective area of the flowing steel. For single phase flow without gas injection, the effective area A_{eff} simply equals the SEN inner cross-section area, A_{SEN} . For multiphase flow, A_{eff} accounts for the extra space taken by the injected gas as follows:

$$A_{eff} = \frac{V_c WT}{Q_{vas} + V_c WT} A_{SEN}$$
(6.6)

where V_c is the casting speed, W and T are the mold width and thickness respectively.

Note that the mass flow rate of the liquid steel can be found by multiplying Q_{SEN} from Eq. (6.2) by the fluid density. This gate-position-based model was used in this work to generate the boundary conditions at UTN top inlet for both the 3-D transient CFD model and the simple analytical equation (AMLE) to investigate mold surface level fluctuations.

6.3.2. Gas Porous-Flow Model

To simulate argon-steel two phase flow, two crucial modeling parameters must first be estimated: the hot gas flow rate entering the nozzle and the resultant initial bubble size distributions after injection, as discussed previously. Most previous work studying argon-steel flow simply converts the measured gas flow rate at standard conditions (STP) (room temperature and pressure) into the hot argon flow rate, using the ideal gas law to account for gas expansion to the molten steel temperature.^[5] However, during the actual process, gas leakage may occur in the system, so the hot argon flow rate entering the nozzle is not always accurately measured. Furthermore, as investigated by Shi and Thomas^[12], bubble size plays an important role in determining the flow pattern in argon-steel two phase simulations, which depends greatly on the gas flow distribution exiting the UTN inner surface.

To obtain these two important parameters, a model for gas flow through nozzle refractory is utilized in the current work to provide a better argon gas inlet boundary condition for the twophase flow simulations. However, simulating multiphase flows is essentially much more difficult compared with single-phase flow calculations, owing partly to the complicated physics from the coupling of different phases (with a high density ratio), and also to the giant uncertainty embedded in the process itself. A systematic procedure has been proposed to model argon-steel two phase flow in nozzle and mold region in the previous work.^[23]

Gas flow through UTN refractory is firstly computed with a porous-flow model, taking into account thermal expansion of the gas and gas temperature-dependent properties by solving for a pressure distribution (via Eq. 6.7), from which the gas velocities are directly calculated via Darcy's law by Eq. (6.8):

$$\nabla \cdot \left(K_D \nabla p \right) = -\frac{RT}{p} \left[\nabla \left(\frac{p}{RT} \right) \cdot \left(K_D \nabla p \right) \right]$$
(6.7)

$$\mathbf{v}_a = -K_D \nabla p \tag{6.8}$$

where K_D is the permeability of the refractory material, p is the argon pressure in the refractory, R is the specific gas constant for argon, and T is the resolved temperature field obtained from a heat transfer calculation of the UTN prior to solving for the pressure distribution via Eq. (6.7). The UTN inner surface gas velocity distribution is then exported as an input condition for the prediction of initial bubble size distribution, combined with an empirical equation from Lee et al.^[25] and a two-stage bubble growth model from Bai and Thomas.^[26] This bubble size distribution is imported into the multiphase flow model to compute the momentum exchange between the discrete bubble phase and the continuous steel phase, which further define flow in the liquid pool and the mold top surface profile. Further details are provided in Chapter 5.

6.3.3. 3-D Transient Eulerian Multiphase Model for SEN / Mold Flows

The computational model of transient turbulent flow is described in this section, including the governing equations, the geometry and mesh of the domain, and the boundary conditions. One of the main reasons to develop the free-surface tracking model is to study the combined effects of rising argon bubbles and flow-rate variations, such as caused by essentially-transient practices such as slide-gate dithering, on the bulk flow pattern and mold top surface motion. Thus, this model system simulates two-phase argon-steel interactions and flow pattern changes in the nozzle and mold bulk regions, in addition to the transient response of the mold top surface level profile. Mass conservation of both the argon-gas and liquid-steel phases is enforced by solving two continuity equations:

$$\frac{\partial (\alpha_a \rho_a)}{\partial t} + \nabla \cdot \left(\alpha_a \rho_a \left(\mathbf{v}_a - \mathbf{v}_g \right) \right) = 0$$
(6.9)

$$\frac{\partial(\alpha_s \rho_s)}{\partial t} + \nabla \cdot \left(\alpha_s \rho_s \left(\mathbf{v}_s - \mathbf{v}_g\right)\right) = 0$$
(6.10)

where α is volume fraction, ρ is density, *t* is time, **v** is the velocity vector field, and subscripts *a*, *s*, and *g*, refer to the argon, steel, and computational grid respectively. Two sets of momentum conservation equations comprise the Eulerian-Eulerian model for argon-steel two phase flows:

$$\frac{\partial (\alpha_a \rho_a \mathbf{v}_a)}{\partial t} + \nabla \cdot (\alpha_a \rho_a \mathbf{v}_a (\mathbf{v}_a - \mathbf{v}_g)) = -\alpha_a \nabla p + \nabla \cdot (\alpha_a \mu_a \nabla \mathbf{v}_a) + K_{as} (\mathbf{v}_s - \mathbf{v}_a) + \alpha_a \rho_a \mathbf{g}$$
(6.11)

$$\frac{\partial (\alpha_s \rho_s \mathbf{v}_s)}{\partial t} + \nabla \cdot (\alpha_s \rho_s \mathbf{v}_s (\mathbf{v}_s - \mathbf{v}_g)) = -\alpha_s \nabla p + \nabla \cdot (\alpha_s \mu_s \nabla \mathbf{v}_s) + K_{as} (\mathbf{v}_a - \mathbf{v}_s) + \alpha_s \rho_s \mathbf{g}$$
(6.12)

The coefficient K_{as} in the above equations defines the interfacial drag forces between argon and liquid steel, which are calculated in Eq. (13).

$$K_{as} = \frac{3}{4} \frac{C_D}{D_b} \alpha_s \rho_s |\mathbf{v}_s - \mathbf{v}_a| \text{ where } C_D = \frac{24}{\text{Re}_b} (1 + 0.15 \text{Re}_b^{0.687}), \text{ Re}_b = \frac{\rho_s |\mathbf{v}_s - \mathbf{v}_a| D_b}{\mu_s}$$
(6.13)

 D_b in Eq. (13) is the bubble diameter, and the Schillar-Naumann drag coefficient C_D is used. ^[37] The argon gas phase and liquid steel phase share the same pressure field, p. Finally, the volume fractions of the liquid steel and argon must satisfy Eq. (14):

$$\alpha_s + \alpha_a = 1 \tag{6.14}$$

6.3.4. Computational Domain and Mesh

Half of the full slide-gate, nozzle and mold region is taken as the computational domain, due to symmetry, as shown in Figure 6.7(a). The mold domain has been divided into two subdomains, an upper 12-cell layer for the dynamic mesh region and a lower fixed-grid region for the remaining bulk of the fluid (2400 mm), as shown in Figure 6.7(a). This division of the fluid bulk is done for computational efficiency, as only the mesh near the meniscus must be smoothed to preserve mesh quality. For best computational efficiency, the upper layer of the mold is assigned to a single processer, while the rest of the domain is split between 5 processors, for parallelization. Thus, slow mesh updating between processors is avoided in each iteration, and computational efficiency is better. A mesh of ~1 million hexahedral cells was adopted for all simulations, as shown in Figure 6.7(b)-(c). Close-ups are shown around the slide-gate region in Figure 6.7(c), and around the free surface region near the SEN and mold in Figure 6.7(c).

6.3.5. Boundary Conditions

The boundary conditions for the dithering process simulations include the UTN inlet, the outlet from the domain bottom, the solidified shell walls, and the mold cavity top surface. For each of these boundaries, a special sub-model was applied, as detailed below.

6.3.5.1. UTN inlet boundary condition with liquid steel flow rate history

Liquid steel flow rate from the flow-rate model is converted to a prescribed-velocity boundary condition at the UTN inlet, based on the cross-sectional area. For steady-state or quasi steady-state simulations without flow rate variations, this inlet steel velocity could be calculated from the casting speed based on the overall mass conservation of the system. However, the dithering of slide-gate causes periodic variations in flow rate/velocity, and this flow rate history is calculated with Eq. (6.2) using the recorded slide-gate position history as an input condition. For multiphase flows with argon injection, argon velocity distributions at the inlet are obtained from the gas porous-flow model,^[23] as previously described in Chapter 2.

6.3.5.2. Convective outlet boundary condition at domain exit

Unlike the steel / argon inlet boundary conditions at the UTN, physical quantities such as pressure and velocities at domain outlet are not known prior to the computation. Thus it is difficult to prescribe an accurate distribution of either liquid steel velocity or pressure at the domain outlet as a boundary condition. Many models in previous work^[3-6,13,27] adopt a pressure boundary condition, where (usually) the ferrostatic pressure at domain exit is estimated and prescribed to approximate the pressure distribution. This boundary condition works well when a fixed no-slip wall boundary condition is used on the mold top surface with no liquid steel penetration. In the current model, the normal component of the average steel velocity at domain exit is enforced to the casting speed using a convective outlet boundary condition.^[38] This condition was successfully used in previous work^[39] at domain exit in transient mold flow simulations, and is adopted here as Eq. (15).

$$\frac{\partial \mathbf{v}^*}{\partial t} + V_C \frac{\partial \mathbf{v}^*}{\partial n} = \mathbf{0}$$
(6.15)

where \mathbf{v}^* is the advected liquid steel velocity at domain outlet and V_C is the fixed casting speed normal to the domain exit plane. In order to ensure the normal component of the spatially averaged liquid steel velocity exiting the domain to be identical with the casting speed, a scaling correction of the normal exiting steel velocities is performed at the end of each iteration:

$$v_n = v_n^* + \frac{Q_T - Q_C}{A_{outlet}}$$
(6.16)

where \underline{v}_n is the normal liquid steel velocity at the domain exit plane, Q_T is the target volumetric flow rate, Q_C is the currently calculated exiting flow rate, and A_{outlet} is the area of the exit plane.

6.3.5.3. Mass and momentum sinks at shell interface boundaries

Liquid steel near the shell interface continuously solidifies into the solid shell, which moves downwards at the constant casting speed. A relatively stable shell profile is maintained in the Eulerian frame of reference. The loss of liquid steel mass and momentum which is accounted for by the mass and momentum sinks as derived in the previous work.^[13] However, in that derivation, the shell interface was assumed to have a planar shape instead of the shape of the square root of distance down meniscus, which is more accurate. A new derivation is given in the current work, which shows that the resulting mass and sink terms are independent of the shape of the shell interface, as long as it is convex. These sink terms are derived and presented in Appendix B.

6.3.5.4. Dynamic boundary condition at mold top surface (slag-steel interface)

Motion of the mold top surface is modeled with the free-surface tracking approach using the FVM dynamic mesh technique addressed in the previous section. The boundary conditions on the slag-steel interface (domain top surface) require no-penetration of fluid through the slag / steel interface (kinematic B.C.) and all forces in equilibrium at free surface (dynamic B.C.). The kinematic boundary condition is automatically satisfied with the calculated surface movements.

At the mold top surface, a liquid flux layer that floats on top of the liquid steel surface adds a roughly constant pressure to the slag-steel interface. The thickness of the top liquid flux layer varies on the surface. Considering the unsteady nature of the slag-steel interface motion, however, it is reasonable to assume that the liquid flux and powder layers do not have enough time to re-distribute during the dithering cycle, so it is reasonable to assume a steady-state slag-layer thickness (pressure) on top of the liquid steel. This assumption has been supported by recent nail-board measurements.^[38] In addition, the curvature of the domain top surface caused by surface velocity variations is balanced by interfacial surface tension. With these assumptions, the dynamic boundary condition is given by:

$$p = p_0 + \rho_{flux} g h_{flux} + \mathbf{n} \cdot \mu \Big[\nabla \mathbf{v} + (\nabla \mathbf{v})^T \Big] \cdot \mathbf{n} + \mathbf{f}_{st} \cdot \mathbf{n}$$
(6.17)

The pressure p_0 in the right-hand side of Eq. (6.17) is atmospheric pressure, (101 kPa), ρ_{flux} is 3000 kg/m³, *g* is 9.81 m/s², h_{flux} is the constant (20-mm) thickness of the flux and powder layers, **v** is the velocity vector at free surface, **n** is the normal unit vector of the local free surface, and **f**_{st} is the surface tension force, which serves two purposes: 1) it enriches the physics at free surface and makes the dynamic boundary condition more accurate, and 2) it dampens local numerical instabilities in the interface tracking solution, which improves convergence.

6.3.6. Free-Surface Tracking Algorithm using Moving-Grid Technique

To track the movement of the top surface of the liquid pool, that is important to level fluctuations, several methods have been used in previous work. A simple method, which converts fluid static pressure to a local level elevation, has been frequently used to estimate the mold level positions.^[6,13,27] However, since it assumes that equilibrium is reached, simulating the dynamics of the mold top surface is beyond the capacity of this method. The VOF method solves a separate advection equation for the phase volume fraction,^[19] which makes it difficult to combine with any other two-phase flow model, such as the Eulerian-Eulerian method for bulk motion of the molten steel and argon gas. The VOF method also requires a very fine mesh (especially in the interface region) and a high order advection scheme to lessen numerical diffusion in order to accurately track the free surface. Finally, interfacial tension is difficult to compute and apply, because the interface is diffuse. Moving grid methods, such as the SPINES method^[32] embedded in the commercial finite-element code, FIDAP, overcome these problems by moving the grid to ensure that the interface lies on the mesh boundary. The SPINES method has been successfully applied to simulate the slag-steel interface shape around a nail^[33] and validated with plant measurements, ^[34] but is prone to convergence problems due to overlapping or misshapen cells.

To resolve these difficulties, a new free-surface tracking method is developed to calculate mold top surface motion, including gravity wave effects. It combines a finite-volume moving-grid algorithm to track the free surface^[35] with a surface tension treatment based on first principles, ^[36] and a dynamic smoothing algorithm for the mesh interior.^[30]

The surface of the flow domain is moved with time in order to keep it at the slag-steel interface. This requires moving the grid surface to satisfy local mass conservation of the fluid mixture, which is also known as a kinematic boundary condition:

$$\left[\left(\mathbf{v}-\mathbf{v}_{g}\right)\cdot\mathbf{n}\right]_{fs}=0$$
(6.18)

where $\mathbf{v}_g = \dot{\mathbf{x}}$, \mathbf{v} is the mixture velocity, \mathbf{v}_g is the velocity of the local free surface, \mathbf{n} is the unit vector normal to the local free surface, and subscript *fs* refers to the top free surface. Integrating

Eq. (6.18) over an arbitrary cell face of a finite-volume discretization, the normal component of fluid mass flow rate exiting the cell face, \dot{m}_{fs} , is compensated by the volume swept by the moving cell face, \dot{V}_{fs} , given as follows ^[35]:

$$\dot{V}_{fs} = -\dot{m}_{fs} / \rho \tag{6.19}$$

The average rate of translation of the local face, \dot{h} , is found from \dot{V}_{fs} divided by the projected area of the cell face. This relation is discretized temporally as follows (6.20).

$$\Delta h = \gamma_{fs} \frac{\dot{V}_{fs} \Delta t}{S_{fs} \mathbf{n} \cdot \mathbf{e}_{fs}}$$
(6.20)

where Δh is the vertical displacement of the face center, γ_{fs} is the relaxation factor (taking the value of 0.2 in the current work), Δt is the time step adopted in the simulation, S_{fs} is the surface area of the local cell face, \mathbf{e}_{fs} is the unit vector in the direction of the local grid motion,

$$\mathbf{e}_{fs} = \frac{\mathbf{x}}{|\mathbf{x}|} \tag{6.21}$$

Average height, which is also defined by the position of the central point in the face, chosen as the "control point"^[35], is updated by:

$$\boldsymbol{x}_{B_i}^{k+1} = \boldsymbol{x}_{B_i}^k + \Delta h \boldsymbol{e}_{fs} \tag{6.22}$$

where \mathbf{r}^{k}_{Bi} is the position of the *i*th face center, at the *k*th flow iteration of the current time step. To find the grid velocity and update the domain coordinates, flow-step iteration continues until Δh for the time step is driven to zero.

Difficulty arises when moving the vertices of the cell faces in the grid or "nodes," x_V , to achieve the desired volume change, defined by control point coordinates x_B , since each node is shared by several mesh faces on the free surface. Movement of the nodes causes volume changes to all sweeping faces sharing that node. To resolve this problem, each node position is updated according to the locations of its neighboring control points, as given by Peri'c.^[35] as follows:

$$\boldsymbol{x}_{V_i}^{k+1} = \boldsymbol{x}_{V_i}^k - \boldsymbol{e}_{fs} \cdot \left(\boldsymbol{x}_{V_i}^k - \sum_{m=1}^n w_m \boldsymbol{x}_{B_i}^k \right) \boldsymbol{e}_{fs}$$
(6.23)

where \mathbf{x}^{k}_{Vi} is the position vector of the *i*th vertex at the *k*th flow iteration, and *w_m* is the weighting fraction of the *i*th control point, found from linear interpolation.

Surface tension is crucial when local small-scale eddies are present in the simulation with large curvatures, and also an important numerical term to stabilize the solution process. So it is important to incorporate the surface tension effect in this free surface tracking algorithm. Calculation (approximation) of the local surface curvature can be achieved in various ways.[] A simple method proposed by Perot and Nallapati [] directly computes the surface tension force at each boundary cell face via :

$$\mathbf{f}_{st} = \frac{1}{A_f} \sum_{e}^{\text{face edge}} \sigma L_e \frac{\left(\mathbf{x}_{fi}^{fc} - \mathbf{x}_{e}^{mid}\right)}{\left|\mathbf{x}_{fi}^{fc} - \mathbf{x}_{e}^{mid}\right|}$$
(6.24)

where σ is the surface tension, L_e is the length of the edge, super-scripts fc and mid beside the position vector **x** denote face center and edge middle point respectively, and the subscripts fi and e indicate neighboring face i and edge e separately.

The variables in Eq. (6.24) are shown in the schematic in Figure 6.8(b). In this 3-D surface mesh, directions of the surface tension force are defined by the vector pointing from the edge center to the center of the corresponding neighboring face that shares the same edge (Figure 6.8a). Due to the simplicity and robustness of Eq. 6.24, together with its natural fit into the iterative solver, this method is adopted in this work to account for the effect of slag-steel surface tension on mold top surface dynamics.

Upon moving the vertices that define the top free surface, the sub-surface cells also need to deform in order to maintain a good quality mesh. The moving boundary (free surface) displacements at each iteration are transferred to each of the underlying nodes in the rest of the grid, according to a diffusion-based smoothing method.^[30]

$$\nabla \cdot \left(\lambda \nabla \mathbf{v}_{g} \right) = 0 \tag{6.25}$$

where λ is a diffusion coefficient, taken as 1 in this work, and \mathbf{v}_g is the spatially-varying grid velocity. The top surface of the dynamic-grid portion of the domain is prescribed according to the above equations. Other boundary conditions on this equation are simply fixed zero velocity at all surfaces of the dynamic-grid domain, which comprises the top layer of the total domain. Eq. (6.25) is solved iteratively using the algebraic multigrid solver in ANSYS-Fluent 14.5, within each (inner) flow iteration of each time step.

6.4. Model Validation

The three models developed in the previous sections are each validated with both analytical solutions and measurements from plant trials or water model experiments before integrating them to investigate the slide-gate dithering process. The porous flow model and Eulerian-Eulerian model have been validated previously in Chapters 2 and 5 respectively.

6.4.1. Validation of Gate-position-based Flow Rate Model

The flow rate model that generates the inlet liquid steel velocity history is validated here with both water model experiments, and measurements from the dithering trials. In addition, the flow-rate model is extended to derive an averaged-mold-level equation (AMLE) model that

converts the predicted flow rate variations in the SEN directly into spatially-averaged variations of the mold top surface.

6.4.1.1. Full-scale water model validation

SEN flow rates were measured as a function of slide-gate opening position in a full-scale water model with a slide-gate control system and compared with the predictions from the gate-position-based flow-rate model. An excellent match with measurements was obtained for gas volume fractions ranging from 0 to 10%, as shown in Figure 6.9. The agreement is significant because the flow-rate model has no fitting parameters. Figure 6.9 also shows that increasing the gas volume fraction, causes the liquid flow rate in the SEN to decrease, for the same gate opening. This effect of gas decreasing the flow rate increases with larger gate openings.

6.4.1.2. Average mold level equation (AMLE) and validation with plant measurements

The spatial-averaged mold level position can be calculated from the flow-rate model output, based on simple mass conservation. For a given casting speed, variations of the UTN inlet liquid steel flow rate are reflected by corresponding variations of the average top-surface level in the mold. These variations can be found by averaging the steel flow-rate history predicted in the SEN over the cross-sectional area of the entire caster. Balancing the flow rate with the average level movement gives:

$$V_l \left(A_l - \frac{\pi}{4} D_o^2 \right) + V_c A_l = Q_{SEN}$$
(6.26)

where V_l (m/s) is the average vertical velocity of the liquid steel surface level in the mold, A_l (m²) is the cross-section area of the mold cavity at the meniscus, D_o (m) is the outer diameter of the SEN, and V_c (m/s) is the casting speed.

Integrating the velocity V_l in Eq. (6.26) gives the mold level h(t):

$$h(t) = \int_{t_0}^t \left(\frac{Q_{SEN}}{\alpha_A A_l} - \frac{V_c}{\alpha_A}\right) dt' + h_0 = \frac{1}{\alpha_A A_l} \int_{t_0}^t Q_{SEN} dt' - \frac{1}{\alpha_A} \int_{t_0}^t V_c dt' + h_0, \qquad \alpha_A = 1 - \frac{\pi D_o^2}{4WT}$$
(6.27)

Integrating Eq. (27) numerically gives:

$$h_{n} = \frac{1}{\alpha_{A}A_{l}} \sum_{i=0}^{n} Q_{SEN}(t_{i}) \Delta t - \frac{1}{\alpha_{A}} V_{c}(t_{n} - t_{0}) + h_{0}$$
(6.28)

where t_n (sec) is the time at time step n, h_n (mm) is the average mold level at time t_n , relative to h_0 at t_0 measured at the start of the time interval, Q_{SEN} (m³/s) is volumetric flow rate from Eq. (6.2), W (m) is the mold width, T (m) is the mold thickness.

As averaged mold level position depends on the integral of the SEN liquid steel flow rate over time, a sudden change of the flow rate more gradually affects the mold level. Similar approaches are u in previous studies where researchers used the mold level fluctuations to calibrate their slide-gate control algorithms.^[41-42] To quantify the fluctuations in mold level, Equation (29) defines the standard deviation or "root mean square" (*rms*) of the mold level fluctuation relative to the mean level (\bar{h}).:

$$h_{rms} = \sqrt{{h'}^2} = \sqrt{\frac{\sum_{i=1}^{N} (h_i - \bar{h})^2}{N}}$$
(6.29)

where h (mm) is the instantaneous mold level; \overline{h} is its time average, and N is the number of sampling points (~1000, taken over a time interval of 10 sec). Larger h_{rms} indicates higher chances of quality problems.

To validate the flow-rate and AMLE models together, their predictions of mold level fluctuations are compared with measurements from several plant trials in Figure 6.10. In most cases, the predictions reasonably match the measurements, which validates the flow-rate model.

However, in the case with a 5.5 mm slide-gate position variation, and 0.9Hz frequency, a huge deviation appears. This is the case where severe mold sloshing was observed. In this case, the averaged mold level fluctuations are caused by gravity waves, so the AMLE prediction based on a mass balance on the flow rate cannot give the real slag-steel interface behavior. Thus, the AMLE model is accurate, unless giant sloshing occurs.

6.4.2. Free Surface Model Validation

To validate the new surface-tracking method, a benchmark problem of 2-D smallamplitude sloshing in a tank^[40] was calculated using the new model. The tank has 1.5m depth, 1.0 m width, and initial free-surface profile given by Equation (25), including a 0.02 m perturbation as pictured in Figure 6.11. The fluid kinematic viscosity is 0.01 m²/s and gravitational acceleration is 1.0 m/s². Two simulations were performed with a 25×40 coarse mesh and 200×80 fine mesh with constant time steps of 0.01 sec for the coarse mesh, and 0.002 sec for the fine mesh. An analytical solution of this small-amplitude sloshing problem was given by Prosperrite.^[29]

$$h(x) = 1.5 + 0.01\sin(\pi(0.5 - x))$$
(6.30)

where h (m) is surface level, x (m) is distance from the left edge of the tank, and surface tension is ignored,.

An excellent match between the computational-model and analytical results is found from the comparison in Figure 6.12. It is worth mentioning that even for a very coarse mesh of 25×40 , a close match between numerical and analytical solutions can still be obtained, even though the grid spacing is much larger than the maximum sloshing magnitude. It is also observed that the fluid viscosity gradually damps the sloshing, but the sloshing frequency remains the same as time progresses. Thus, the new free-surface tracking model developed in the current work is an ideal approach to model top-surface mold-level fluctuation behavior in continuous casting, especially for problems involving multiphase flow.

6.5. Quasi-steady Results with Single-Phase Flow

Before performing a simulation of transient behavior such as dithering, the simulation is first run for ~10 sec (physical time) until a quasi-steady flow field is established. Instantaneous results using two different turbulence models, the $k-\omega$ model^[43] and the Detached Eddy Simulation (DES) model,^[44] are compared in this section. The $k-\omega$ model resembles the standard $k-\varepsilon$ two-equation model for Reynolds averaged Navier-Stokes (RANS) simulations, but with improved prediction in flows with adverse pressure gradient or flow separations. The DES performs Large Eddy Simulation (LES) in the fluid bulk region to capture the local instantaneous turbulent eddies, and adopts a RANS wall function close to the no-slip wall to resolve the details of the turbulent boundary layer fluctuations with less computational effort than LES.^[44]

The liquid steel flow pattern in the center plane under quasi-steady conditions before dithering is shown in Figure 6.13, for both the k- ω (Figure 6.13a) and DES models (Figure 6.13b). A double-roll flow pattern is observed with both models. The snapshot from the transient DES model captures the turbulent variations due to the swirling jet, but the time-averaged flow pattern is similar to that using the *k*- ω model. The corresponding free surface shape and velocities results from these two models are compared in Figure 6.14 (a) and (b). The *k*- ω model generates the smooth top surface profile expected from a time-averaged method, while DES gives an instantaneous surface shape with expected variations. Both simulations give a lower surface level and higher surface velocity around the quarter mold region. Near the SEN and the narrow face, the top surface rises and becomes more varied, while the surface velocity is lower.

Figure 6.15(a) shows the zoomed-in deformed shape of the mold top surface from the DES model. The vertical (z-direction) length scale is stretched by 5 times, for easier visualization. Small vortices and wrinkles appear, especially near the SEN outer perimeter and meniscus. The mold level rises close to the SEN, due to the impingement of the liquid steel stream flowing towards the SEN. Velocity vectors along the free surface are plotted in Figure 6.15(b). Recirculation regions are observed near both sides of the SEN, which cause dimples (depressions) to form on the free surface. In extreme conditions of left-right asymmetric flow, these depressions can grow into vortices that entrain mold flux.^[45]

6.6. Dithering Results with Single-Phase Flow

This section presents the transient flow pattern and evolution of the slag-steel interface for the dithering process conditions given in Table I Cases 1 and 2. Both a half-mold and a fullmold simulation were performed using the k- ω model. These cases used the same nozzle and slide-gate used to generate the flow rate curves in Figure 6.9. A total flow rate of 1 SLPM argon was injected into the UTN, corresponding to at most 1% gas volume fraction in hot condition. This is small enough that single phase flow was assumed in the current simulation. The predicted mold level fluctuations are compared with plant measurements. Then, a parametric study is then conducted to investigate the average mold level fluctuations during typical dithering processes.

6.6.1. Case 1 – Mold Sloshing Study

Simulation results for the dithering process conditions that caused mold sloshing in the plant trials, (0.9 Hz dithering frequency and 14 mm dithering stroke), are shown in Figure 6.16 (a)-(e). Approximately 5 dithering cycles were simulated with the half-mold model (starting from 0 sec after stable flow was established), and the cycle from 3.0 sec to 4.1 sec is presented in Figures 6.16 and 6.17. At time 3.0 sec at the start of a dithering cycle, the strength of the liquid steel jet starts to increase. The mold top surface level is higher close to the SEN, and lower at the narrow face. At time 3.3 sec, the jet grows stronger, and the recirculation velocity around the upper roll is accelerated. During this time, the mold surface level near the SEN decreases, while the surface level at mold narrow face increases. The slag-steel interface is relatively flat by this time. At time 3.6 sec, the established jet starts to decrease in strength, due to the upstream closing of the slide-gate. At this moment, the surface level at mold narrow face reaches its peak. At time 3.8 sec, the jet is weakest, and the mold surface level near the SEN keeps increasing, while the level at mold narrow face continues to decrease. When the time reaches 4.1 sec, the surface level at the SEN has risen to its highest point, and begins to drop, while the level at the narrow face starts to increase from its lowest point. At the same time, the liquid steel jet strength starts to increase, and the next dithering cycle begins.

Throughout each dithering cycle, the liquid steel jet leads a wobbling path towards the mold narrow face due to the flow rate variation, and the flow pattern in mold region remains double-roll. The duration of this large scale eddy is long enough that the change of inlet liquid steel flow rate does not alter the flow pattern. The recirculation velocity in the upper roll is accelerated and slowed down periodically. When the dithering frequency matches the mold natural frequency, as in this simulation, the increase of the jet strength accelerates the rising velocity of the mold surface level at narrow face. In this manner, the kinetic energy increase

from the jet outweighs the energy dissipated by the viscous effects. Thus mold sloshing becomes energized and magnified in every dithering cycle.

The 3-D motion of the mold top surface is shown in Figure 6.17 (a)-(e), where the contours indicate the local level. Dark regions indicate high surface levels, while red regions have lower levels. From time 3.0 sec to 3.3 sec, surface level drops near the SEN, and increases near the narrow face. At time 3.3 sec, the surface levels are almost even. Then level at the narrow face starts increasing, from the increased momentum of the liquid steel jet flowing up the narrow face. At time 3.6 sec, surface level reached its maximum at the narrow face. For the next 0.5 sec, a ~35mm-high wave flows across the top surface towards the SEN, finally producing a maximum surface level at the SEN at time 4.1 sec. This time coincides with the beginning of the next dithering cycle, which causes the process to repeat in an amplified manner during the next dithering cycle.

A full-mold simulation of dithering case 1 is also performed with results of flow pattern evolution during one dithering cycle at the mold center plane shown in Figure 6.18. Similar flow patterns and mold top surface shape are found between the left and right half of the mold, which approximate the flow patterns simulated using the half-mold computational domain. Thus, the left-right symmetry assumption for the previous half-mold simulation has been validated. The mold level fluctuations indicated by the mold quarter point during 8 dithering cycles from the full mold simulation are plotted in Figure 6.19 and compared with the measured mold level at that point from the eddy-current sensor. A reasonable match is found, which validates the model system used in this transient simulation.

To evaluate the evolution of the mold level profile during a typical slide-gate dithering cycle in more detail, mold surface profiles at the center line between mold broad faces are shown

211

in Figure 6.20 at different times during the 1.1 sec cycle. At time 6.5 sec, mold level near the SEN has risen to its highest point, and has dropped to its lowest position at the narrow face, at both left and right sides of the mold. After 0.3 sec (at time 6.8 sec), the level decreases near the SEN, and increases at the narrow face, making the profile relatively flat, especially near the narrow face. At time 7.1 sec, level position near the SEN drops to its lowest point, and rises to its highest point at the narrow face. This profile is flipped from that at time 6.5 sec, which shows the lowest-frequency standing wave in the half-mold region, corresponding to a (1,0) mode by Eq. (1). At time 7.3 sec, mold level position increases near the SEN and decreases at the narrow face. At time 7.6 sec, level profile returns to that at the beginning of the cycle, with the highest position near SEN and lowest at narrow face, which matches well with the profile at time 6.5 sec.

Mold level profiles taken at the same time during each of the 8 dithering cycles, when the maximum level is found at the narrow face, are compared in Figure 6.21. As time progresses, the sloshing magnitude increases, although at a diminishing rate. The maximum level difference across the mold increases from ~20 mm in the 5th cycle (4.9 sec), and reaches ~60 mm by the 12^{th} cycle (12.7 sec). The level profiles (sloshing amplitude) at time 11.6 sec and 12.7 sec match very well with each other, indicating that a periodically "stable" state is almost reached for the mold sloshing. The left and right sides of the mold do not exactly match, however, which indicates that lower frequency oscillations with lower magnitude are also occurring at the same time.

6.6.2. Case 2 – High-Frequency Dithering

Flow-pattern results with a higher dithering frequency of 1.4 Hz, (Case 2) are shown in Figure 6.22 for a typical dithering cycle. A double-roll flow pattern is still predicted in the mold

region, but no mold sloshing is found with this dithering frequency. Variation of the mold top surface during the same dithering cycle is shown in Figure 6.23(a)-(d), corresponding to four snapshots in Figure 6.22(a)-(d). As expected, the surface in this case has much smaller fluctuations than with the critical 0.9 Hz dithering frequency. At time 2.4 sec, the mold top surface is relatively flat, with a depression near the SEN along the mold transverse (thickness) direction. After 0.25 sec (2.65sec), as the SEN flow rate increases, the mold top surface velocity increases and the previous depression disappears. Instead, the local mold level rises and form a peak on the top surface. This traveling wave moves away from the SEN towards the narrow face, with a velocity of ~0.1 m/s, as plotted in Figure 6.23(c). However, this traveling wave moves only a short distance before it diffuses, and a new top surface depression occurs at approximately the same location as in Figure 6.23(a), and one dithering cycle has been completed.

6.6.3. Parametric Studies using AMLE

Average mold level fluctuations are calculated by combining the SEN flow rate model in Eq. (6.2) with the AMLE model in Section 6.4.2.2 to perform a parametric study varying: casting speed, mold width, and dithering stroke. Standard conditions are 1.0 m/min speed, 1842mm width, 1.0 Hz frequency, and 14-mm stroke and other conditions are given in Table I Case 2. For dithering practices which do not activate the gravity-wave sloshing mechanism, mass conservation dominates the mold level fluctuations, so the AMLE model should be reasonably accurate, as discussed previously. Five dithering strokes were modeled, 6, 8, 10, 12, and 14 mm, at three different casting speeds, 0.6, 1.0, and 1.4 m/min, (25, 40 and 55 inches per min). Results are presented in Figure 6.24(a). Note that the actual peak-to-valley mold level fluctuations are

around 3 times of the mold level *rms* values calculated, as shown on the right axis in Figure 6.24(b).

6.6.3.1.Effect of dithering stroke and frequency

The effect of dithering stroke is shown in Figure 6.24(a) for different casting speeds. Liquid steel flow rate changes during dithering according to the dithering stroke. This causes the average mold levels to vary, while casting speed is constant. Increasing dithering stroke increases these flow-rate variations, which in turn increases the mold level fluctuations. Specifically, increasing dithering stroke from 6 mm to 14 mm increases flow rate variations by ~140 percent, causing level fluctuations to increase from ~0.25 mm to ~0.55 mm (*rms*) for low (0.6 m/min) casting speed, from ~0.3 mm to ~0.7 mm for medium (1.0 m/min) casting speed, and from ~0.35 mm to ~ 0.85 mm for high (1.4 m/min) casting speed. Increasing dithering frequency causes average level fluctuations to decrease.

6.6.3.2.Effect of casting speed and width

The effect of casting speed is shown in Figure 6.24(b) for different dithering strokes. As casting speed increases, the flow rate variations through the UTN increase as well. At lower casting speeds, the *rms* of mold level fluctuation is less than 0.7 mm, even with a high dithering stroke. When casting speed increases to 1.4 m/min, the *rms* of mold level fluctuation increases to ~ 0.8 mm for higher dithering strokes. The increase of mold level fluctuation (*rms*) is almost uniform when casting speed increases from 0.6 m/min to 1.0 m/min and from 1.0 to 1.4 m/min, for each of the dithering strokes. Specifically, average mold fluctuations increase from ~ 0.7 mm to ~ 0.85 mm (*rms*) when increasing steady-state casting speed from 1.0 to 1.4 m/min (40 to 55 ipm), with a 14 mm dithering stroke.

Next, a matrix of parameters with 4 different mold widths (1.1, 1.3, 1.5 and 1.8 m) and 6 different casting speeds (0.6, 0.9, 1.0, 1.1, 1.3, 1.4 m/min) are adopted for the parametric studies on mold level fluctuations (*rms*). Figure 6.24(c) shows the effects of both casting speed and mold widths on average mold level fluctuation. It is observed that for each of mold width, mold level fluctuation increases with increasing casting speed, which is consistent with the trend shown in Figure 6.24(b). It is also shown in Figure 6.24(c) that at a given casting speed, increasing mold width causes mold level fluctuations to increase, owing to the increase in the mean flow rate. As shown by the level fluctuation map in Figure 6.24(c), the largest mold level fluctuation occurs at the highest casting speed (1.4 m/min) with the narrowest mold width (1.1 m), and the least level fluctuation occurs with the lowest casting speed (0.6 m/min) and the largest mold width (1.8 m).

6.7. Dithering Results with Multiphase Flows

The flow behavior and modeling procedure for dithering with argon injection are both much more complicated than with single-phase flows. The argon flow rate entering the liquid steel stream is first calculated using the porous flow model presented in the previous sections, and the initial bubble size is estimated combining an experimental correlation for active site density and a two-stage model to predict initial bubble sizes, taking into account the liquid steel flow rate variations during slide-gate dithering. Then the predicted liquid steel and argon gas flow rates were imported into the SST k- ω , two-phase flow model as inlet boundary conditions for these two phases. The moving-grid free surface tracking algorithm computes the mold top surface motion and deformation during the dithering process.

6.7.1. Slide-Gate Position and Liquid Steel Flow Rate History

The plant recorded slide-gate position and the measured mold level histories are plotted in Figure 6.25. The window in Figure 6.25 shows a ~40 sec time interval for the dithering simulation, in which the dithering stroke switches from 14 mm to 7 mm, while the dithering frequency is kept constant. The sampling frequency during the measurements is 40 Hz (one sampling point per 0.05 sec). Based on slide-gate position and estimated gas flow rate, the SEN liquid steel flow rate between time 160 sec and 200 sec can be calculated via Eq. (6.2), as shown in Figure 6.26(a). It is observed that the change of liquid steel flow rate (mean nozzle velocity) during one dithering cycle is approximately between 50% (for 7 mm dithering stroke) to 100% (for 14 mm dithering stroke). This huge variation of the liquid steel flow rate causes large-scale flow pattern variations in both the nozzle and mold regions and also affects the gas injection rate (with constant back pressure) as well as the resultant distribution of the initial bubble sizes.

6.7.2. Gas Flow Rate and Initial Bubble Size Estimation

As shown in Table I, both the gas injection pressure (back-pressure) and the gas flow rate were measured and recorded in the plant. However, the measured gas flow rate usually is greater than that entering the liquid steel stream in the nozzle, due to leakage. The argon flow rate entering liquid steel is calculated using the porous-flow model presented in Section 6.3.3, for each data point on the liquid steel flow rate curve. To avoid a computationally-intensive 3-D transient porous-flow simulation, a solution of the following 1-D temperature and pressure distribution along the radial direction through the UTN refractory wall is calculated:

$$\frac{1}{r}\frac{d}{dr}\left(r\frac{dT}{dr}\right) = 0 \tag{6.30}$$

$$\frac{d^2p}{dr^2} + \frac{1}{r}\frac{dp}{dr} = \left(\frac{1}{T}\frac{dT}{dr} - \frac{1}{K_D}\frac{dK_D}{dr}\right)\frac{dp}{dr} - \frac{1}{p}\left(\frac{dp}{dr}\right)^2$$
(6.31)

where *r* is the radial position (ranging between 0.04 to 0.075 m), *T* is the temperature, *p* is the gas pressure, and K_D is the refractory permeability.

After evaluating the temperature, from the simple analytical solution to Eq. (6.30), pressure can be solved, considering the one-way coupling. Eq. (6.31) has an analytical solution for constant permeability, which is derived in Appendix D. However, when the gas viscosity varies with temperature, K_D also varies with temperature, and numerical solutions are needed. In the present work, gas viscosity varies with temperature following the same correlation as in Chapter 2. Eq. (31) is solved iteratively using a TDMA algorithm, with the non-linear source term on the right-hand-side explicitly updated.

Because SEN liquid steel flow rate changes with the slide-gate position as time evolves, the pressure inside the nozzle also varies with time, based on Bernoulli's equation (6.2). This change of local pressure inside the nozzle affects the diffusion of argon through the UTN refractory, as well as the resultant bubble sizes. For each slide-gate position, the liquid steel flow rate is first calculated following Eq. (6.2), and then the local pressure is calculated using Bernoulli's equation. The calculated pressure inside the nozzle is then used as the boundary condition for the porous flow model to predict the gas velocity distributions across the refractory wall. The calculated gas flow-rate history is shown in Figure 6.26(b), together with the predicted initial bubble size history and argon volume fraction history. As seen in Figure 6.26(b), the mean argon flow rate is around 6 SLPM, (4% gas), which is much less than the measured gas flow rate of 20 SLPM (corresponding to $6x10^{-4}$ kg/s as mass flow rate or 14% gas). This suggests that the possible gas leakage occurs during the process, and about 70% of the gas has leaked away. It is worth pointing out that quantifying the gas flow rate entering the liquid steel and the gas leakage requires the porous-gas flow model prediction. The calculated gas flow rate is then imported into

the multiphase flow simulations with slide-gate dithering effect as the inlet boundary condition for the argon phase.

The effect of liquid steel cross-flow velocity variation on the initial bubble size is more complicated. When the cross-flow liquid steel velocity increases, local pressure inside the nozzle drops, as indicated by Bernoulli's equation (6.2) and the gas flow rate increases due to the increased pressure drop across the nozzle wall. This encourages bubbles to grow larger before detaching (for the same number of active sites). On the other hand, the increase of bulk steel flow rate in the nozzle tends to increase the number of active sites on the UTN inner surface, which could reduce the gas flow rate through each active site, leading to smaller bubbles. Moreover, as the liquid steel cross-flow velocity increases, the "sweeping" effect that shears off the bubbles also increases, leading to a decrease in the initial bubble size. The net effect of the liquid steel cross-flow velocity variation on the resulting initial bubble size is shown in Figure 6.26(b). It is seen that increasing the cross-flow liquid steel velocity decreases the initial bubble size, and a 50% decrease is found in the initial bubble size as the liquid steel cross-flow velocity increases from 0.8 to 1.56 m/s. Argon volume fraction history based on the superficial flow rates of both the liquid steel and argon phases is also calculated and plotted in Figure 6.26(c). It is observed that both flow rates increase and decreases together, with a maximum gas volume fraction (~5%) occurring when the argon flow rate reaches its minimum.

The average initial bubble size distribution over these 16 dithering cycles is shown in Figure 6.27. However, because the Eulerian-Eulerian model does not track the trajectories of the bubbles, the changing initial bubble size during dithering cannot be directly incorporated in the model. Instead, an average bubble size is adopted to approximate the typical distribution of sizes dispersed in the nozzle and mold flows. This approximation is reasonable because the difference

in the terminal bubble velocities in the downward-flowing liquid steel nozzle tends to mix together large and small bubbles that formed at different times. The Sauter-mean bubble diameter over these dithering cycles (from Figure 6.27) is calculated as 2.75 mm, which is used in the subsequent transient multiphase simulations as a constant.

6.7.3. Initial Solution for Dithering Simulation with Multiphase Flow

Before the dithering simulation starts, the quasi steady-state flow pattern in the SEN and mold region is solved with the model system. The flow patterns for both liquid steel and argon gas at the mold center plane are shown in Figure 6.28. Under this gas injection rate and gas volume fraction, the liquid-steel flow pattern is still "double-roll", with gas bubbles rising close to the SEN. The mold top surface, shown in Figure 6.28, has a depression of ~ 10 mm in the high-velocity region near mold quarter point, similar to that in single-phase flow. The mold top surface shape is shown in Figure 6.29(a) (liquid steel velocity) and 6.29(b) (argon gas volume fraction). The maximum surface steel velocity occurs between the mold quarter point and the narrow face. The argon gas is distributed mainly between the mold quarter point and SEN, with a maximum gas fraction of ~5% found close to the SEN.

6.7.4. Case 3 Multiphase Flow Results

Results are presented in Figure 6.30 to 6.32 for a typical low-frequency (0.4 Hz) dithering cycle with large (14 mm) stroke. This simulation used a flat top wall and a simple pressure conversion method, described below, to predict top-surface level profile, for comparison purposes. Figure 6.30 displays the liquid steel velocity distribution at SEN port exit, projected to the horizontal direction. Most of the liquid exits the port bottom, owing to the downward

momentum of the nozzle flow, and the typical oversized port area, which is 2.0 times the nozzle bore area. In addition, a strong swirling is observed. This swirling phenomenon is caused by the blocking effect of the slide gate. Asymmetric liquid steel flow is produced below the partially-open slide gate in the SEN. This downward velocity along the north side of the SEN generates a strong counterclockwise swirl at the SEN bottom, (viewed looking into the west port) and this swirl is sustained through the port exit into the liquid pool in the mold. Argon gas velocity and volume fraction distributions through the dithering cycle (170 to 172 sec) are shown in Figure 6.31. The argon enters the mold cavity mainly from the upper left region of the port, where it is volume fraction exceeds 45%, and generally from around the center of the port, where it is 10% - 20%. This is because the steel flow exits more from the bottom and outside right regions of the port, and the low-density, low-momentum argon flow concentrates in the opposite regions.

The transient fluctuations of the jet during the example dithering cycle are illustrated in Figure 6.32. The jet generally enters the mold cavity at $\sim 20^{0}$ downward, which is slightly steeper downward than the port angle, as expected.^[8] In addition, the strong swirling flow sends a jet upwards from the upper port, creating a "nose" in the jet profile entering the mold cavity. This swirl and nose is most prominent during the time of maximum flow rate, which is experienced midway through the dithering cycle, at ~171 sec. The increase in flow rate, which occurs during the first 3 frames of this figure, causes the jet in the mold cavity to "wobble". This generates fluctuations in the flow pattern, in addition to an overall increase in velocity in the mold cavity. Later in the cycle, (eg. 172sec), as the flow rate decreases, the jet tends to straighten out.

The behavior of the argon gas in the mold region is shown in Figure 6.33. Periodic variations of the liquid steel flow due to the dithering causes accompanying variations in the argon gas fraction of the flow, which injects high-concentration gas "pockets" into the mold at

the same frequency of ~2.5sec. These gas pockets contain ~25-50% gas. The high buoyancy of these gas pockets causes them to leave the jet and float directly to the top surface, which takes ~5sec. Because this period is roughly double the injection frequency, there are usually two gas pockets observed in the mold cavity at any instant in time. One gas pocket is just emerging from the port, and another is still rising to the surface from the previous cycle. The gas pocket elongates as it rises, producing the higher-velocity plume next to the SEN in the frames of Figure 6.33. Periodic change of liquid steel flow rate during dithering process also affects the mold level profile and fluctuations at the top surface. In this simulation using the pressure method, a flat wall was imposed at the top surface, and the mold level is calculated as:

$$\Delta h = \frac{p - p_0}{\rho_L g} \tag{6.32}$$

where Δh is the mold level deviation, p is the pressure at top surface, and p_0 is the pressure corresponding to the reference mold level used in determining Δh . ρ_L is the liquid density, and gis the gravitational acceleration (9.8067 m/s²). Displacement of the liquid slag layer is neglected in this equation, because the entire layer was judged to be thin enough to simply rise and fall with the steel surface profile variations. A comparison between the predicted mold level during the simulated 40 sec of dithering and the measured unfiltered mold level data is shown in Figure 6.34.

Excellent agreement between the predicted and measured mold level is found in Figure 6.34, with both the level fluctuation magnitude and phase matching very well with each other. Both the prediction and measurement show smaller mold level fluctuations when the dithering stroke is decreased. Specifically, the measured level fluctuations decrease from 4.7mm to 2.7mm. The mold level fluctuation frequency roughly matches the dithering frequency of 0.4 Hz. As the casting speed stays constant, and the amplitude and frequency of the level fluctuations relate directly to the dithering stroke and frequency, simple mass conservation over the whole system is the main phenomenon controlling surface level variations during this dithering process, especially when dithering stroke is large. Secondary phenomena involve the feedback caused by reinforcement of the flow in the upper recirculation zones, gravity waves, and mold level oscillations, which will be investigated in future work with this model to optimize the dithering process.

6.7.5. Case 4 Multiphase Flow Results

The flow pattern evolution during one low-frequency (0.4 Hz) dithering cycle is shown in Figure 6.35 (a)-(d) for small (7 mm) stroke. At time 35.3 sec, the steel flow rate starts to increase, with the mold level near SEN slightly higher than that at the narrow face. At time 36.6 sec, the steel flow rate increase to its maximum, with a mold level profile similar to that at time 35.3 sec. The steel flow rate decreases at time 37.3 sec, with the highest mold level position found at the narrow face. At time 37.8 sec, flow rate of steel reduces to the minimum, with the highest mold level position near the SEN, and the lowest at narrow face, matching well with the flow pattern and surface profile found at time 35.3 sec. It is observed that sloshing occurs in the half-mold region in this low-frequency (0.4 Hz) dithering case. The natural frequency for the mold sloshing with a (1,0) mold can be found as 0.91 Hz from Eq. (1) based on the mold width, which is more than twice larger than the dithering frequency of 0.4 Hz. Thus during one dithering cycle, mold sloshing occurs twice, as can be seen in Figure 6.35.

Evolution of the gas volume fraction is shown in Figures 6.36(a)-(d). High gas concentration areas are found in front of the SEN port and in the region near SEN above the

nozzle port. The gas fraction field evolves during slide-gate dithering, with the high concentration region above the nozzle port moving towards the top surface.

Both the simulated and measured mold levels during the 9-sec simulation are presented and compared in Figure 6.37, together with the liquid steel rate history. A close match is found for the spatial-averaged mold level between the calculations using the averaged mold level equation (AMLE) during this event and the simulation results with the top surface tracking model. This helps validate the numerical model. The measured level history at the mold quarter point exhibits the same trend as the averaged mold level, with similar fluctuation magnitude (3-4 mm). Level fluctuations monitored at mold quarter point from the simulation results also follow the measured quarter point level closely, but reveal a more wiggling variation with time compared with the measurements. The frequency of the level wiggling is ~0.9 Hz, which matches with the natural frequency of this mold for a (1,0) sloshing mode. This indicates that the wiggling is likely caused by mold sloshing, which has also been observed in Figure 6.34-6.35. This discrepancy between the measurements and simulation could be explained by the possible filtering operation of the mold level signal during the measurements. Backlash might also exist in the slide-gate system during this small-amplitude dithering (with a 6mm stroke), which would further reduce the flow rate variations and the level fluctuations. It is also shown in Figure 6.37 that the level change signal is delayed (shifted forward in time) by ~0.6 sec after the liquid steel flow rate signal. This is the response time, which theoretically corresponds to one quarter of a dithering cycle, for the mold level to adapt its position to the velocity variation, which is determined by the frequency of the flow rate signal.

6.8. Conclusions

This work applies plant measurements and computational models to investigate the effect of slide-gate dithering on mold level fluctuations. Two different systems of models have been developed, validated individually, and applied to capture the different physics occurring in this complicated real-world problem. Both a complex 3-D transient turbulent flow CFD model and a simple analytical model have been constructed to investigate the transient variations of flow pattern change, evolution of the slag-steel interface and average mold level fluctuations during the dithering of slide-gate for different operating conditions. A novel free-surface tracking model with a dynamic mesh technique was created utilizing the dynamic mesh feature of the commercial FVM CFD package ANSYS-FLUENT. From the results of both models, the following conclusions can be drawn:

- The new free-surface tracking methodology is proven to be a promising approach to model both transient and steady state behavior of the slag-steel interface in terms of accuracy, computational efficiency and applicability to complex multiphase flow problems including severe sloshing;
- 2. Severe mold sloshing occurs when the dithering frequency matches the natural frequency of the mold, which is determined only by the mold dimensions;
- The critical sloshing frequency is 0.9Hz for the 1840mm mold width studied here and decreases with increasing width;
- 4. The mechanism of sloshing and its detailed behavior is revealed by the simulation results, where the gravity wave periodically is energized by the increase of liquid steel jet momentum during the dithering cycle, resulting in the magnification of surface waves and mold level fluctuations;

224

- 5. The average mold level fluctuation can be calculated accurately using a simple analytical model developed in this work, as long as there is no severe sloshing;
- 6. Increasing casting speed during the dithering increases the magnitude of the mold level fluctuations. Specifically, average mold level fluctuations increase from ~0.7 to ~0.9 mm (rms) or ~2.0 to ~2.4 mm (peak to valley) when increasing steady-state casting speed from 1.0 to 1.4 m/min (40 to 55 ipm), with a 14 mm dithering stroke. Thus, casting speed should be restricted when casting wide slabs (eg. 1840 mm or 72.5 inch), especially with high dithering stroke;
- 7. Increasing dithering stroke also increases the mold level fluctuations. Specifically, average mold level fluctuations increase from ~0.3 mm to ~0.7 mm (rms), or ~0.8 mm to ~2.0 mm (peak to valley) when dithering stroke changes from 6 mm to 14 mm, for the casting speed of 1.0 m/min (40 ipm), but ~0.4 to ~0.8 mm (rms) or ~1.0 to ~2.4 mm (peak to valley) for the casting speed of 1.4 m/min (50 ipm). Thus, dithering strokes larger than 12 mm should be avoided for casting speeds above 1.0 m/min (40 ipm) for wide slabs.

6.9. Tables and Figures

		Case 1	Case 2	Case 3	Case 4
Casting Speed	(m/min)	1.02	1.02	1.02	1.02
Mold Width	(mm)	1842	1842	1829	1829
Mold Thickness	(mm)	254	254	254	254
SEN Port Angle	(deg.)	15	15	15	15
Submerged Depth	(mm)	143	143	203	203
Dithering Stroke	(mm)	14.0	7.0	14.0	7.0
Dithering Frequence	cy (Hz)	0.9	1.4	0.4	0.4
Gas Inj. Pressure	(psi)	8	8	19	19
Gas Flow Rate	(SLPM)	1	1	20	20

Table I. Casting Conditions for Dithering Trials



Figure 6.1. Dithering trials with mold level measurements



Figure 6.2. Influence of dithering frequency on mold level fluctuations



Figure 6.3. Schematic of sloshing mode indices



Figure 6.4. Mold natural frequency curves


Figure 6.5. Model system for dithering process simulations



Figure 6.6. Top View of Gate and SEN Inner Bores



Figure 6.7. Domain geometry and mesh setup



Figure 6.8. 3-D Boundary surface mesh layout and surface tension calculation



Figure 6.9. Comparison of flow rate model predictions with water model measurements



Figure 6.10. Comparison between measured and AMLE-predicted mold level fluctuations



Figure 6.11. Schematic for small sloshing in a tank

Figure 6.12. Validation of the free-surface tracking model



Figure 6.13. Comparison of simulated mold center plane flow patterns by different turbulence models



Figure 6.14. Comparison of simulated mold top surface morphologies by different turbulence models



Figure 6.15. Zoomed-in of free surface close to SEN calculated using DES (length magnified by 5 times along z-axis), with free surface mesh, (a), and steel velocities, (b).



Figure 6.16. Transient liquid steel flow pattern evolution during one dithering cycle for the mold sloshing case 1



Figure 6.17. Mold top surface behavior during one dithering cycle (Case 1)



Figure 6.18. Full mold flow evolution during a dithering cycle with mold sloshing (Case 1)



Figure 6.19. Comparison between measured and calculated mold level at quarter point (Case 1)



Figure 6.20. Mold level evolution during a typical dithering cycle (Case 1)



Figure 6.21. Maximum sloshing amplitude over 8 dithering cycles (Case 1)



Figure 6.22. Flow pattern evolution at dithering frequency 1.4 Hz (Case 2)



Figure 6.23. Mold top surface evolution (Case 2)



Figure 24. Effects of dithering stroke, casting speed and mold width on flow rate variation and mold level fluctuation (f=1.0 Hz)



Figure 6.25. Slide-gate position history and measured mold level



Figure 6.26. Computed histories (*f*=0.4 Hz) of liquid steel flow rate, (a), argon flow rate and volume fraction (b) and initial bubble size (c)



Figure 6.27. Initial bubble size distribution during slide-gate dithering



Figure 6.28. Initial flow fields for multiphase dithering case



Figure 6.29. Initial solution of steel velocity and argon fraction at mold top surface



Figure 6.30. Liquid steel velocity distribution at SEN port exit, axis in (m)



Figure 6.31. Argon velocity and concentration distribution at SEN port exit, axis in (m)



Figure 6.32. Evolving of liquid steel velocity distribution at center plane during dithering, axis in (m)



Figure 6.33. Argon gas fraction variation during dithering process, showing periodic gas pockets, axis in (m)



Figure 6.34. Measured and predicted mold level histories using both the pressure method and the moving mesh model (m)



Figure 6.35. Steel flow patter evolution during one dithering cycle (*f*=0.4 Hz)



Figure 6.36. Argon volume fraction evolution during one dithering cycle (f=0.4 Hz)



Figure 6.37. Liquid steel flow rate and mold level signals

6.10. Reference

- B.G. Thomas, "Fluid Flow in the Mold", Chapter 14 in Making, Shaping and Treating of Steel, 11th Edition, Vol. 5, Casting Volume, A. Cramb, ed., AISE Steel Foundation, Pittsburgh, PA, Oct., 14.1-14.41, 2003.
- [2] J. Knoepke, M. Hubbard, J. Kelly, R. Kittridge, and J. Lucas, "Pencil Blister Reduction at Inland Steel Company", *Steelmaking Conf. Proc.*, 77, ISS, Warrendale, PA, 1994, pp. 381-388.
- Y. Wang and L. Zhang, "Transient Fluid Flow Phenomena during Continuous Casting: Part I – Cast Start", *ISIJ International*, 50(12), 2010, pp. 1777-1782.
- Y. Wang and L. Zhang, "Transient Fluid Flow Phenomena during Continuous Casting: Part II – Cast Speed Change, Temperature Fluctuation, and Steel Grade Mixing", *ISIJ International*, 50(12), 2010, pp. 1783-1791.
- [5] X. Huang and B.G. Thomas, "Modeling of Transient Flow Phenomena in Continuous Casting of Steel", *Canadian Metallurgical Quarterly*, 37(304), 1998, pp. 197-212.
- [6] R. Liu, B.G. Thomas and J. Sengupta, "Simulation of Transient Fluid Flow in Mold Region during Steel Continuous Casting," *Modeling of Casting, Welding, and Advanced Solidification Processes (MCWASP) XIII*, (Styria, Austria, June 17-22, 2012), 2012.
- [7] R. Liu, B.G. Thomas, L. Kalra, T. Bhattacharya and A. Dasgupta, "Slidegate Dithering Effect on Transient Flow and Mold Level Fluctuations", *Proc. of AISTech Conf.*, 2013, pp. 1351-1364.
- [8] H. Bai and B.G. Thomas, "Effects of Clogging, Argon Injection and Continuous Casting Conditions on Flow and Air Aspiration in Submerged Entry Nozzles", *Metallurgical and Materials Transactions B*, 32(4), 2001, pp. 707-722.

- [9] K.G. Rackers and B.G. Thomas: "Clogging in Continuous Casting Nozzles", in 78th Steelmaking Conference, vol. 78, (Nashville, TN, April 2-5, 1995), ISS, Warrendale, PA, 1995, pp. 723-34.
- [10] R. Sánchez-Perez, R. D. Morales, M. Díaz-Cruz, O. Olivares-Xometl and J. Palafox-Ramos, "A Physical Model for the Two-Phase Flow in a Continuous Caster Mold", *ISIJ International*, 43(5), 2003, pp. 637-646.
- [11] L. Zhang, S. Yang, X. Wang, K. Cai, J. Li, X. Wan, and B.G. Thomas, "Investigation of Fluid Flow and Steel Cleanliness in the Continuous Casting Strand", *Metallurgical and Materials Transactions B*, 38B(1), 2007, pp. 63-83.
- [12] T. Shi and B.G. Thomas: "Effect of Gas Bubble Size on Fluid Flow in Continuous Casting Mold", Continuous Casting Consortium at University of Illinois at Urbana-Champaign, Report, 2001.
- [13] Q. Yuan, B.G. Thomas and S. P. Vanka, "Study of Transient Flow and Particle Transport during Continuous Casting of Steel Slabs, Part 1. Fluid Flow", *Metallurgical and Materials Transactions B*, 35B(4), 2004, pp. 685-702.
- [14] R. Liu, W. Ji, J. Li, H.F. Shen and B.C. Liu, "Numerical Simulation on Transient Flow Patterns of Upper Rolls in Continuous Slab Casting Moulds", *Steel Research International*, 79, 2008, pp. 50-55.
- [15] R. Chaudhary, B.G. Thomas and S.P. Vanka, "Effect of Electromagnetic Ruler Braking (EMBr) on Transient Turbulent Flow in Continuous Slab Casting using Large Eddy Simulations," *Metallurgical and Materials Transactions B*, 43B(3), 2012, pp. 532-553.

- [16] R. Singh, B.G. Thomas and S.P. Vanka, "Effects of a Magnetic Field on Turbulent Flow in the Mold Region of a Steel Caster", *Metallurgical and Materials Transactions B*, 44B(5), 2013, pp. 1201-1221.
- [17] Z. Liu, B. Li, M. Jiang and F. Tsukihashi, "Modeling of Transient Two-Phase Flow in a Continuous Casting Mold using Euler-Euler Large Eddy Simulation Scheme", *ISIJ International*, 53(3), 2013, pp. 484-492.
- Z. Liu, B. Li, and M. Jiang, "Transient Asymmetric Flow and Bubble Transport Inside a Slab Continuous-Casting Mold", *Metallurgical and Materials Transactions B*, 45, 2014, pp. 675-697.
- [19] C.W. Hirt and B.D. Nichols, "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries", *Journal of Computational Physics*, 39, 1981, pp. 201-225.
- [20] L. Kalra, A. Dasgupta, W. Umlauf and K. Zheng, "Dither a Necessary Evil Phase 1", AISTech 2012 Conf., Atlanta, US.
- [21] R. M. McDavid and B. G. Thomas, "Flow and Thermal Behavior of the Top Surface Flux/Powder Layers in Continuous Casting Molds", *Metallurgical and Materials Transactions B*, 27, 1996, pp. 672-685.
- [22] A. Dusgupta, L. Kalra, K. Zheng and W. Umlauf, "Dither, a Necessary Evil Phase 2", *Prof. of AISTech 2013 Conf.*, Pittsburgh, Pennsylvania, 2013, pp. 2427-2438.
- [23] R. Liu and B.G. Thomas: "Model of Gas Flow through Upper Tundish Nozzle Refractory and Initial Bubble Size", *Proc. of AISTech 2012 Steelmaking Conf.*, Atlanta, GA, 2012, pp. 2235-2246.
- [24] G. Houghton, A.M. McLean, and P.D. Ritchie: "Mechanism of formation of gas bubblebeds", *Chemical Engineering Science*, 1957, 7(1–2), pp. 40-50.

- [25] G. Lee, B.G. Thomas and S. Kim, "Effect of Refractory Properties on Initial Bubble Formation in Continuous-casting Nozzles", *Metal and Materials International*, 2010, vol. 16(3), pp. 501-506.
- [26] H. Bai and B.G. Thomas: "Bubble Formation during Horizontal Gas Injection into Downward Flowing Liquid", *Metallurgical and Materials Trans. B*, 2001, vol. 32B(6), pp. 1143-1159.
- [27] R. Liu, B.G. Thomas, B. Forman and H. Yin, "Transient turbulent flow simulation with water model validation and application to slide gate dithering", *AISTech Iron Steel Technol. Conf. Proc.*, 2012, Atlanta, GA, pp 1317.
- [28] H. Lamb, *Hydrodynamics*, Dover Publication, 6th Edition, 1932.
- [29] A. Prosperetti, "Motion of Two Superposed Viscous Fluids", *Physics of Fluids*, 1981, 24(7), pp 1217.
- [30] Fluent ANSYS Inc., User Manual, 2011.
- [31] H. Oertel, L. Prandtl, et.al, "Prandtl's Essentials of Fluid Mechanics", *Springer*, ISBN 0387404376. pp. 163–165.
- [32] M.S. Engleman, *FIDAP Theoretical Manual*, Fluent, Inc., 2001.
- [33] B. Rietow and B.G. Thomas, "Using Nail Board Experiments to Quantify Surface Velocity in the CC Mold", Proc. of AISTech 2008 Steelmaking Conf., Pittsburgh, PA, (2008).
- [34] R. Liu, J. Sengupta, D. Crosbie, S. Chung, M. Trinh and B.G. Thomas, "Measurement of Molten Steel Surface Velocity with SVC and Nail Dipping during Continuous Casting Process", *TMS Annual Meeting*, San Diego, CA, 2011, pp. 51-58.

- [35] S. Muzaferija and M. Peric, "Computation of Free-Surface Flows using the Finite-Volume Method and Moving Grids", *Numerical Heat Transfer, Part B: Fundamentals*, 1997, 32(4), pp. 369-384.
- [36] B. Perot and R. Nallapati, "A Moving Unstructured Staggered Mesh Method for the Simulation of Incompressible Free-Surface Flows", *Journal of Computational Physics*, 2003, 184, pp. 192-214.
- [37] L. Schiller and Z. Naumann, "A Drag Coefficient Correlation", Z. Ver. Deutsch. Ing., 77, 1935, pp. 318-320.
- [38] A. Sohankar, C. Norberg, and L. Davidson: Int. J. Num. Meth. Fluid., 1998, vol. 26, pp. 39–56.
- [39] R. Chaudhary, C. Ji, B.G. Thomas and S.P. Vanka, "Transient Turbulent Flow in a Liquid-Metal Model of Continuous Casting, Including Comparison of Six Different Models", *Metallurgical and Materials Transactions B*, 2011, 42(5), pp. 987-1007.
- [40] B. Ramaswamy and M. Kawahara, "Arbitrary Lagrangian–Eulerian Finite Element Method for Unsteady Convective Incompressible Viscous Free Surface Flow", Int. J. Numer. Meth. Fluids, 1987, 7, pp 1053.
- [41] R. de Keyser, "Improved Mould-Level Control in a Continuous Steel Casting Line", *Control Engineering Practice*, 1997, 5(2), pp 231.
- [42] Q. Liu and W. Wang, "A Mold Level Mechanism Model in Continuous Casting with Application to Suppression of Limit Cycle", *ACTA AUTOMATIC SINICA*, 2002, 28(3), pp 156.
- [43] F.R. Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", AIAA Journal, 1994, 32(8), pp 1598.

- [44] P.R. Spalart, W.H. Jou, M. Strelets and S.R. Allmaras, "Comments on the Feasibility of LES for Wings and on the Hybrid RANS/LES Approach", Proc. of the 1st AFOSR Int. Conf. on DNS/LES, Arlington, TX, 1997.
- [45] S.M. Cho, G.G. Lee, S.H. Kim, R. Chaudhary, O.D. Kwon and B.G. Thomas, "Effect of Stopper-rod Misalignment on Asymmetric Flow and Vortex Formation in Steel Slab Casting", *TMS Annual Meeting*, Seattle, WA, 2010, pp. 71-77.

CHAPTER 7. CONCLUSIONS

This work established a modeling procedure to systematically investigate essentially transient argon-steel two phase flows in the nozzle and mold regions of continuous slab casters. The procedure combines four different mathematical models to study different aspects of physics during the transient multiphase flow process, including 1) a porous-flow model (or pressure-source model) to calculate argon gas pressure and velocity distributions through the upper tundish nozzle (UTN) refractory, 2) (semi-) analytical models to predict the liquid steel flow rate in the nozzle both in the stopper-rod system (stopper-position-based model) and in the slide-gate system (gate-position-based model), respectively, 3) multiphase flow models (both the Eulerian-Eulerian and the Eulerian-Lagrangian models) to compute the flow fields for both argon gas and liquid steel phases, and 4) a finite-volume-based free-surface tracking algorithm with a moving grid technique to capture the mold top surface evolution during transient events. All the models are validated by both analytical solutions and experiment measurements, and then applied to simulate practical processes. Summaries regarding utilizations of each of these models in different applications are presented in the following sub-sections.

7.1. Model of Gas Flow Through Porous Refractory Applied to UTN

Two models, a porous-flow model and a pressure-source model, are developed to simulate gas flow inside the porous refractory considering the geometry of the refractory, gas thermal expansion and temperature-dependent gas viscosity. The models are validated by both the 1-D porous gas flow analytical solutions derived in a cylindrical coordinate system, and observations from a bubbling experiment. The porous-flow model is then applied to study the gas flow through a UTN porous refractory with different gas injection pressures and refractory permeability under different joint sealing conditions, with the following major findings:

- A critical gas injection pressure exists to overcome the bubbling threshold for gas to enter the liquid steel stream. A one-way flow pressure boundary condition has been developed and applied to prevent unphysical reverse gas flows into the refractory, which puts the model into practical use.
- Design of the nozzle geometry and location/size of the gas injection slits have a huge impact on the gas velocity distribution at UTN inner surface. Bubbles exit preferentially through locations near the injection slits, and the low pressure regions, because of the higher pressure gradients around those areas.
- Significant gas escape occurs if the joint leaks. This leakage ratio could reach 100% depending on the leaking area at the joint and the nozzle/slit designs. For the specific design studied in this work, the leakage reaches as high as 80%, however, with a double-slit gas delivery.
- Increasing the gas injection pressure naturally increases the gas flow rate, under the same casting conditions (e.g. casting speed, mold width, tundish level). If leakage occurs, increase in the injection pressure decreases the leakage ratio, for the double-slit design adopted in this work. Variation in specific permeability of the refractory, however, does not affect the pressure distribution, but changes the gas velocity in a linear manner.

Besides providing argon gas flow rate as boundary condition for the subsequent two-phase flow simulations in the nozzle/mold regions, the initial bubble size can also be predicted following a

procedure developed in this work, combining models and correlations developed by previous researchers. Therefore these models provide

7.2. Measurements of Molten Steel Surface Velocities and Effects of Stopper-Rod Movements on Transient Multiphase Flow in Continuous Casting

The simple nail dipping approach is adopted to measure molten steel surface velocities. A new correlation is proposed based on a least square regression of previous simulation results by Rietow and Thomas, to quantify liquid steel velocities using the measured diameter and height difference of the solidified lump around the nail perimeter. This correlation has been validated by the SVC measurements during the same plant trials. Computational models for tow-phase flow simulations are validated by the nail dipping measurements, and applied to simulate the flow pattern evolutions during a transient event with multiple stopper-rod movements which leads to defects in the final product. The following conclusions are drawn:

- In general, surface velocity increases with increasing casting speed, and/or decreasing gas volume fraction (mean gas volume fractions). Reverse surface steel flows (velocities pointing away from the SEN to the narrow face) are revealed under higher gas fractions (>~10%) by numerical simulation results and plant measurements, using both nail dipping and the SVC device.
- Mold width affects steel surface velocities in a complicated manner. Increasing mold width increases the steel throughput under the same casting speed and increases the jet traveling distance to reach the narrow face. The net effect revealed in the current work suggests an increase of surface velocity with increasing mold width.

253

• The stopper-position-based model provides a reasonably accurate flow rate history which serves as an input for the multiphase flow simulation. Flow filed evolution during the transient event reveals a significant disturbance on the mold top surface during the stopper-rod movements (or the "declogging" process), which further leads to slag entrainment and defect formation.

7.3. Particle Transport and Deposition in A Turbulent Square Duct Flow with An Imposed Magnetic Field

DNS simulation is performed to compute the instantaneous velocity field of the continuous phase in a turbulent square duct flow ($\text{Re}_{\tau} = 360$), and the Lagrangian particle tracking is used to calculate the particle trajectories via a one-way coupling approach. The particle logic is developed and integrated in the in-house code, CUFLOW, and implemented on graphic processing units (GPU) card (Tesla C2075). The following findings regarding particle dispersion and deposition in the duct flow are revealed:

- In both MHD and non-MHD flows, particles tend to accumulate in the saddle regions between turbulent eddies, but away from the secondary eddy centers. In the near wall region, particles gather preferentially in regions with a positive −τ_p∇**u**_f : ∇**u**_f value.
- Particle deposition patterns a very different between flows with and without MHD effects. A significant reduction of particle deposition rate is found on the duct walls perpendicular to the imposed magnetic field, especially near the wall center region.
- Deposition rate increases with particle Stokes number under the same flow conditions. The overall particle deposition rate in non-MHD square duct flows is about 2-5 times higher than that in the MHD flows, for corresponding particle Stokes numbers.

7.4. Computational and Experimental Studies of Argon-Steel Flows in A Continuous Caster Mold

Multiphase flows in the mold region are investigated by both numerical simulations and plant measurements using the nail-board method. Three sets of casting parameters were chosen for the investigation, with different casting speeds, mold widths and gas injection rates (volume fractions based on the superficial velocities). The porous-flow model developed previously in this work is used to calculate the gas velocity distributions in the nozzle refractory, and the initial bubble sizes are predicted for the simulations. Both Eulerian-Eulerian and the Eulerian-Lagrangian models are adopted in this work to predict the argon-steel two phase flow patterns. The simulated steel surface velocities and mold level profiles are then compared with those from the nail-board measurements, with reasonable matches obtained. The following conclusions are drawn:

- The calculated gas superficial velocities and resulting initial bubble sizes deepen the understanding of the gas injection process, which are also the two key parameters for the sub-sequent multiphase flow simulations.
- For the single phase liquid steel flows with a wide strand (1732 mm), low-frequency jet wobbling is predicted by both URANS model and the DES model. However, a relatively small amount of gas injection reduces this wobbling and increases the left-right symmetry of the mold flow.
- Reverse flows are successfully predicted by the numerical simulations which a relatively large gas injection rate, which is consistent with the previous finds from nail dipping measurements. Parametric studies with different bubble sizes indicate

that for cases with relatively low gas volume fractions, smaller bubbles tend to decreases molten steel surface velocities, while for cases with reverse flows under a relatively high gas injection rate, steel flow is not very sensitive to a certain rage of bubble sizes (3-8 mm).

• Both the simple pressure method and the moving-grid free surface tracking algorithm for mold level calculations are validated by the nail board measurements. It has been proved that this pressure method can be well used in quasi-steady state mold level calculations.

7.5. Modeling Transient Multiphase Flow and Top Surface Dynamics in Continuous Caster Mold during Slide-Gate Dithering

The complete model system developed in this work is applied to investigate the flow pattern evolution and mold top surface motion during slide-gate dithering with and without argon injection. The novel free-surface tracking algorithm with the moving-grid technique is developed and validated for the mold top surface calculation in the multiphase flow case. Five scenarios are chosen for the investigation, with the following findings:

- The new free-surface tracking methodology was proven to be a promising approach to model both transient and steady state behavior of the slag-steel interface in terms of accuracy, computational efficiency and applicability to complex multiphase flow problems including severe sloshing.
- Severe mold sloshing occurs when the dithering frequency matches the natural frequency of the mold, which is determined only by the mold dimensions. The mechanism of sloshing and its detailed behavior is revealed by the simulation results,

where the gravity wave periodically is energized by the increase of liquid steel jet momentum during the dithering cycle, resulting in the magnification of surface waves and mold level fluctuations.

• The average mold level fluctuation can be calculated accurately using a simple analytical model developed in this work, as long as there is no severe sloshing. Increasing casting speed during dithering increases the magnitude of the mold level fluctuations. Increasing dithering stroke also increases the mold level fluctuations.

CHAPTER 8. RECOMMENDATIONS FOR FUTURE WORK

The current work has established a model system to investigate transient multiphase flows and top surface dynamics in continuous caster molds with improved accuracy. Based on this work, future work can be carried on in two directions: further computational tool developments and novel practical applications. Recommendations of the future work regarding these two directions are briefly addressed in this chapter.

- For future computational tool development:
 - 1) The initial bubble size procedure described in Chapter 2 and Chapter 5 consists of three parts: a porous-flow model, an active site distribution density correlation (Lee, 2010), and a two-stage bubble formation model (Bai, 2001). The experiment correlation can be improved by more parametric studies on the factors that affect the bubble sizes. The two-stage bubble formation semi-analytical model could also be improved with a more fundamentally-established criterion for bubble detachment at active sites, which could possibly expand the usability of this model in lower cross-flow velocity regions, with better accuracy.
 - 2) The finite-volume-based moving-grid free surface tracking algorithm can be implemented in the GPU-based in-house code, CUFLOW, coupled with LES to capture instantaneous free surface behavior. The accuracy of this model partly depends on the accuracy of the turbulence models used in the calculation. LES has shown to be a more accurate model over the URANS models especially in simulating essentially transient processes. Regarding the way to move the grid vertices (nodes) on the top surface in CUFLOW after calculating the control point displacements at the end of each time step, a better method could be adopted to

improve the method used in Chapter 6 (Perić, 1997). An implicit least square approach can be used to obtain more robust and more accurate node displacement solutions, which also couples with the velocity update schemes in CUFLOW (e.g. Adams-Bashforth scheme) in a more natural way.

- 3) Evolution of bubble size in the caster nozzle and mold regions can be studied using population-balance-type models (e.g. MUSIG in CFX and PBM in ANSYS Fluent), which might deepen the understanding of the two-phase fluid dynamics occurring in the caster.
- For novel practical applications in continuous casting:
 - 1) The two (semi-) analytical flow rate models, a stopper-position-based model and a gate-position-based model, were requested to be integrated into the flow control systems in ArcelorMittal Inc. to predict clogging index and to help with mold level control practice. Effect of actuator backlash on the flow rate change can also be incorporated in these models. The mold level results from model parametric studies can be used to help the mold level control system reduce level fluctuations and improve product quality.
 - 2) The porous-flow model combining with the initial bubble size prediction procedure contributes to the study of bubble size control. Previous work has studied optimal bubble sizes in the mold region for steel cleanliness concerns. Parametric studies can be carried out using the bubble size prediction procedure to determine the argon injection rate (by controlling the back-pressure during gas injections) under certain casting conditions to optimize the bubble sizes, which is beneficial to the practical operation.

- 3) Lagrangian particle tracking module developed in CUFLOW in Chapter 4 in this work can be directly applied to the caster geometry to study inclusion transport and entrapment in the mold. The code can also be used to simulate argon bubble motion and interaction with liquid steel in the mold region.
- 4) The well-established free-surface tracking model introduced in Chapter 6 can be coupled with level control algorithms used in the actual slide-gate system (the control algorithm is integrated into Fluent UDFs to couple the CFD calculations) to simulate mold top surface behavior under a specific set of operation parameters, especially with argon injections. Thus the plant mold level control algorithms can be tested an improved based on the simulation results, which provide much more details regarding level fluctuations and surface velocity variations, with and without slide-gate dithering.

APPENDIX A.

ANALYSIS OF FLUID FLOW AND HEAT TRANSFER IN THE LIQUID FLUX LAYER IN THE MOLD-SHELL GAP – INCLUDING THE EFFECTS OF GAP SIZE, VARYIG SHELL TEMPERATURE AND FLUX VISCOSITY

A.1. Introduction

Liquid flux flow and heat transfer behavior in the gas between caster mold wall and solidified steel shell is important to the understanding of mold slag consumption and related defects formation. Analytical models are derived to solve for the liquid flux flow rate in the gap of a continuous caster under different scenarios with an arbitrary gap size and a temperature-dependent flux viscosity to determine the final mold flux consumption rate. This appendix derives and documents an iterative procedure to solve for the flux velocity distributions and resulting flow rate in the gap between the mold and solidified steel shell inside a continuous caster. Each successive attempt elucidates a key aspect of the problem and is incorporated into the final solution. The first solution step considers the gap as two parallel plates and the fluid viscosity as a constant. This case demonstrates how the flow rate changes as the flux progresses down the caster and led to the inclusion of mass balance in the second attempt. The second step considers the changing viscosity and mass balance of the liquid flux between two parallel plates to determine the flux flow rate. Step two was than generalized for an arbitrary gap size and temperature distribution to determine the final mold flux consumption rate.

A.2. Scenario I: Liquid Flux Flow between Parallel Plates with Constant Viscosity

This case considers the situation of flow between two parallel plates at various temperatures across the gap. The pressure in the system is considered linear and equal to the weight of the steel. The viscosity is assumed constant for a given temperature and the velocity profiles of the fluid are calculated. If we consider the varying viscosity in the flux flow in the mold-shell gap, the following governing equations apply for a 2D dimension case:

Continuity equation:

$$\frac{\partial V_x}{\partial x} + \frac{\partial V_z}{\partial z} = 0 \tag{A.1}$$

X-momentum equation:

$$\frac{\partial V_x}{\partial t} + V_x \frac{\partial V_x}{\partial x} + V_z \frac{\partial V_x}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \frac{1}{\rho} \left(\frac{\partial}{\partial x} \left(\mu \frac{\partial V_x}{\partial x} \right) + \frac{\partial}{\partial z} \left(\mu \frac{\partial V_x}{\partial z} \right) \right)$$
(A.2)

Z-momentum equation:

$$\frac{\partial V_z}{\partial t} + V_x \frac{\partial V_z}{\partial x} + V_z \frac{\partial V_z}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \frac{1}{\rho} \left(\frac{\partial}{\partial x} \left(\mu \frac{\partial V_z}{\partial x} \right) + \frac{\partial}{\partial z} \left(\mu \frac{\partial V_z}{\partial z} \right) \right) + g$$
(A.3)

Energy equation:

$$\rho C_P \left(\frac{\partial T}{\partial t} + V_x \frac{\partial T}{\partial x} + V_z \frac{\partial T}{\partial z} \right) = k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial z^2} \right)$$
(A.4)

The above governing equations are simplified by applying the following conditions:

 Steady state fluid flow and heat transfer. Casters are known to continue running consistently for a long period of time and the energy input in the system does not change with time. Therefore the transient term is assumed zero.

- 2) The temperatures at the mold wall and flux/shell interface are fixed. This is a simplification that is removed in Case II and III. It is used to see how the flux velocity profile changes within the gap.
- 3) Flow in the gap is approximated as the flow between parallel plates. It is easy to consider parallel plates. Two gap sizes are used to show how the flow is modified by a change in the gap. This simplification is removed in Case III.
- 4) Fully developed flow between the gap and the thermal "fully developed" condition is reached. With this simplification we have removed changes in temperature and flow in the z-direction. This is simplification is removed in Case II and III.
- 5) Incompressible Newtonian fluid for liquid flux.
- 6) Linear pressure distribution at shell-flux interface equal to the weight of the steel. This simplification gives us a starting point for the pressure distribution in the flow. The pressure distribution is solved in Case II and Case III.

Based on the assumptions above, the governing equations in A.1-A.4 can be simplified into the following form:

1) Continuity equation simplifies by condition 4:

$$\frac{\partial V_x}{\partial x} = 0 \tag{A.5}$$

and further $V_x = 0$ based on the impermeable wall boundary condition.

The x-momentum equation is reduced by conditions 1, 4, 5, and continuity:

$$0 = -\frac{1}{\rho} \frac{\partial P}{\partial x} \tag{A.6}$$

indicating that the pressure is not changing across the gap.
The z-momentum equation:

$$0 = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \frac{1}{\rho} \left(\frac{\partial}{\partial x} \left(\mu \frac{\partial V_z}{\partial x} \right) \right) + g$$
(A.7)

Energy equation:

$$0 = k \left(\frac{\partial^2 T}{\partial x^2}\right), \text{ or } \frac{\partial^2 T}{\partial x^2} = 0$$
 (A.8)

The simplification of this problem is all based on the "fully developed flow" assumption. If the flow is not fully developed, then the z-momentum equation cannot be further simplified into a 1-D case, thus the analytical solution will be unavailable. The same applies for the energy equation. The sketch of the domain is shown in Figure 1:



Figure A.1 Domain sketch and linear pressure distribution assumption

For thermally fully developed flow, as assumed, the temperature will always be linearly distributed across the gap. So the solution to the energy equation follows:

$$T = T_{fsol} + \frac{T_{shell} - T_{fsol}}{H_{liq}} x$$
(A.9)

 T_{fsol} is the temperature fixed at the solid liquid flux interface and is constant. T_{shell} is the temperature fixed on the shell-flux interface and is also a constant. Insert the temperature solution to the viscosity formulae, resulting in:

$$\mu = \mu_0 \left(\frac{T_0 - T_{fsol}}{T - T_{fsol}} \right)^n = \mu_0 \left(\frac{T_0 - T_{fsol}}{T_{fsol} + \frac{T_{shell} - T_{fsol}}{H} x - T_{fsol}} \right)^n$$
(A.10)

Define:

$$\Delta \rho = \rho_{steel} - \rho_{slag}$$

$$\Delta T = T_{shell} - T_{mold}$$
(A.11)

So the viscosity is now a function of the location across the gap. The z-momentum equation can be simplified as:

$$\frac{d}{dx}\left(\mu\frac{dV_z}{dx}\right) = \frac{dP}{dz} - \rho g \tag{A.12}$$

The pressure gradient could be approximated as the linearly distributed ferrostatic pressure acting on the shell by the molten steel. The assumption is that the locally varying pressure enforcing the global continuity is very small in magnitude comparing with the ferrostatic pressure. So the pressure acting on the shell-flux interface will still be close to the linear distribution of the ferrostatic pressure. The actual pressure is calculated in Case II and III.

Integrating the z-momentum equation:

$$\mu \frac{dV_z}{dx} = \left(\rho_{steel} - \rho_{slag}\right)gx + C_0 = \Delta\rho gx + C_0 \tag{A.13}$$

where C_0 is a integration constant.

Integrate the z-momentum equation again:

$$V_{z} = \int \left(\frac{\Delta \rho g x}{\mu} + \frac{C_{0}}{\mu}\right) dx = \frac{1}{\mu_{0} \left(T_{0} - T_{fsol}\right)^{n}} \int \left(\Delta \rho g x + C_{0}\right) \left(T_{mold} + \frac{\Delta T}{H} x - T_{fsol}\right)^{n} dx + C_{1} (A.14)$$

With all the details of this derivation neglected, the final form of this solution follows:

$$V_{z} = \frac{1}{\mu_{0} \left(T_{0} - T_{fsol}\right)^{n}} \left[\frac{\Delta \rho g H}{(n+1)\Delta T} x \left(\frac{\Delta T}{H} x + T_{mold} - T_{fsol}\right)^{n+1} - \frac{\Delta \rho g H^{2}}{(n+1)(n+2)\Delta T^{2}} \left(\frac{\Delta T}{H} x + T_{mold} - T_{fsol}\right)^{n+2} + C_{1} \right] + C_{0} \frac{H}{(n+1)\Delta T} \left(\frac{\Delta T}{H} x + T_{mold} - T_{fsol}\right)^{n+1} \right] + C_{1}$$
(A.15)

The integration constants will be determined by the boundary condition:

At *x*=0, the velocity is zero, due to the no-slip wall B.C.

$$\frac{1}{\mu_0 \left(T_0 - T_{fsol}\right)^n} \left[C_0 \frac{H}{(n+1)\Delta T} \left(T_{mold} - T_{fsol}\right)^{n+1} - \frac{\Delta \rho g H^2}{(n+1)(n+2)\Delta T^2} \left(T_{mold} - T_{fsol}\right)^{n+2} \right] + C_1 = 0$$
(A.16)

At *x*=*H*, the velocity is the casting speed, which is also due to the no-slip wall B.C.

$$\frac{1}{\mu_{0}\left(T_{0}-T_{fsol}\right)^{n}} \begin{bmatrix} C_{0} \frac{H}{(n+1)\Delta T} \left(T_{shell}-T_{fsol}\right)^{n+1} - \frac{\Delta \rho g H^{2}}{(n+1)(n+2)\Delta T^{2}} \left(T_{shell}-T_{fsol}\right)^{n+2} \\ + \frac{\Delta \rho g H^{2}}{(n+1)\Delta T} \left(T_{shell}-T_{fsol}\right)^{n+1} \end{bmatrix} + C_{1} = V_{casting}$$
(A.17)

Further re-organization of the equations above leads to:

$$\frac{1}{\mu_{0}\left(T_{0}-T_{fsol}\right)^{n}} \begin{bmatrix} C_{0} \frac{H}{(n+1)\Delta T} \left(\left(T_{shell}-T_{fsol}\right)^{n+1} - \left(T_{mold}-T_{fsol}\right)^{n+1} \right) - \\ \frac{\Delta \rho g H^{2}}{(n+1)(n+2)\Delta T^{2}} \left(\left(T_{shell}-T_{fsol}\right)^{n+2} - \left(T_{mold}-T_{fsol}\right)^{n+2} \right) + \\ \frac{\Delta \rho g H^{2}}{(n+1)\Delta T} \left(T_{shell}-T_{fsol}\right)^{n+1} \end{bmatrix} = V_{casting} \quad (A.18)$$

From the equation above, C_0 is evaluated as:

$$C_{0} = \frac{(n+1)\Delta T (T_{0} - T_{fsol})^{n} \frac{V_{casting} \mu_{0}}{H} - \Delta \rho g H (T_{shell} - T_{fsol})^{n+1}}{(T_{shell} - T_{fsol})^{n+1} - (T_{mold} - T_{fsol})^{n+1}} + \frac{\Delta \rho g H}{(n+2)\Delta T} \frac{(T_{shell} - T_{fsol})^{n+2} - (T_{mold} - T_{fsol})^{n+2}}{(T_{shell} - T_{fsol})^{n+1} - (T_{mold} - T_{fsol})^{n+1}}$$
(A.19)

And C_1 is:

$$C_{1} = \frac{1}{\mu_{0} \left(T_{0} - T_{fsol}\right)^{n}} \left[\frac{\Delta \rho g H^{2}}{(n+1)(n+2)\Delta T^{2}} \left(T_{mold} - T_{fsol}\right)^{n+2} - C_{0} \frac{H}{(n+1)\Delta T} \left(T_{mold} - T_{fsol}\right)^{n+1} \right]$$
(A.20)

Due to the complexity of these expressions, the numbers for the constants are calculated using the scientific computing commercial package, Matlab, given the casting parameters, and the final form of the velocity profile depends on the boundary conditions as well as the fluid thermal/mechanical properties. A table for the fluid properties and corresponding velocity profiles is shown below:

Case Index	Interface Temperature (<i>C</i>)	Shell Temperature (<i>C</i>)	Liquid Flux Gap Size (<i>mm</i>)	C_0	C_{I}
А	1145	1250	1	-4.3490	0.0
В	1145	1200	1	25.2105	0.0
С	1145	1190	1	39.8361	0.0
D	1145	1180	1	63.6191	0.0
E	1145	1170	1	108.3879	0.0
F	1145	1160	1	219.9215	0.0
G	1145	1200	3	-74.6234	0.0
Н	1145	1160	3	-9.7197	0.0
Ι	1145	1250	3	-84.4765	0.0

Table I. Case Parameters (Scenario I.)

Case A:
$$V_{z} = 0.0019x(105000x+0)^{2.15} - 5.7625 \times 10^{-9}(105000x+0)^{3.15}$$
$$-1.8171 \times 10^{-7}(105000x+0)^{2.15} + 0$$
(m/s)

Case B:
$$V_z = 0.0036x(55000x+0)^{2.15} - 2.1002 \times 10^{-8}(55000x+0)^{3.15} + 2.0109 \times 10^{-6}(55000x+0)^{2.15} + 0$$
(m/s)

Case C:
$$V_{z} = 0.0044x(45000x+0)^{2.15} - 3.1373 \times 10^{-8}(45000x+0)^{3.15} + 3.8836 \times 10^{-6}(45000x+0)^{2.15} + 0$$
 (m/s)

Case D:
$$V_z = 0.0057x(35000x+0)^{2.15} - 5.1862 \times 10^{-8}(35000x+0)^{3.15} + 7.9743 \times 10^{-6}(35000x+0)^{2.15} + 0$$
 (m/s)

Case E:
$$V_{z} = 0.0080x(25000x+0)^{2.15} - 1.0165 \times 10^{-7} (25000x+0)^{3.15} + 1.9020 \times 10^{-5} (25000x+0)^{2.15} + 0$$
 (m/s)

Case F:
$$V_{z} = 0.0133x(15000x+0)^{2.15} - 2.8236 \times 10^{-7}(15000x+0)^{3.15} + 6.4321 \times 10^{-5}(15000x+0)^{2.15} + 0$$
(m/s)

Case G:
$$V_{z} = 0.0109x(18333x+0)^{2.15} - 1.8902 \times 10^{-7} (18333x+0)^{3.15} - 1.7857 \times 10^{-5} (18333x+0)^{2.15} + 0$$
 (m/s)

Case H:
$$V_z = 0.04x(5000x+0)^{2.15} - 2.5412 \times 10^{-6}(5000x+0)^{3.15} - 8.5282 \times 10^{-6}(5000x+0)^{2.15} + 0$$
 (m/s)

Case I:
$$V_{z} = 0.0057x(35000x+0)^{2.15} - 5.1862 \times 10^{-8}(35000x+0)^{3.15} - 1.0589 \times 10^{-5}(35000x+0)^{2.15} + 0$$
(m/s)

From **Case A** to **Case F**, the corresponding velocity profiles across the mold-shell gap are plotted in Fig. 2 below:



Figure A.2. Velocity profiles across **1 mm** mold-shell gap at different shell temperatures

The analytical solutions above are compared with numerical simulations for validation and implementation. The computational details for the numerical models are listed in Table 2. The boundary condition for the shell-flux interface adopts a no-slip moving wall condition, and the B.C. for the mold-flux interface uses a no-slip stationary wall. The laminar flow model is used instead of any turbulence models, due to the low Re number of ~4 calculated previously. The domain dimensions are **1 mm x 800 mm**, with a pressure boundary condition applied at both upper and lower domain boundaries. The mesh adopts **30x800** cells. The heat transfer process is coupled with fluid flow, and the fluid viscosity is a function of local temperature. Previous derivation of the analytical solution shows that under a thermally "fully developed" condition, the temperature distribution across the gap follows a linear distribution, and the heat conductivity constant is not involved. The viscosity then changes with temperature across the gap. The fluid flow is then affected by the varying viscosity. The varying viscosity across the gap is modeled via implementing a User Defined Function (UDF) in ANSYS Fluent v6.3. The cases simulated include **Case A**, **Case B**, and **Case F** in previous section. For the case with a shell-flux interface temperature of 1200 C, the velocity profile is plotted in Fig.3.



Figure A.3. Comparison of analytical solution with numerical simulation

For this case (shell temperature being 1200 C, gap size 1 mm), the analytical solution matches nicely with the result from numerical model. Numerical simulations of Cases G, H and I are performed and results are shown in Fig. 4 below.



Figure A.4. Velocity profiles across **3 mm** mold-shell gap at different shell temperatures – analytical and numerical solutions

It is observed from Fig. 4 that the velocity profiles from both computational model and analytical model match better with a lower shell temperature. Higher shell temperatures will generate thinner fluids, thus more upward flow. The uncertainty of upward flow temperature is the main reason for the mismatch between analytical model and the computational ones. So it is expected that the case with lower shell temperature will result in better match with the simulation. Conclusions for Case I:

- 1) Analytical solutions based on the assumption of hydraulic/thermal fully developed flows match reasonably with numerical model results, suggesting:
 - a. Fully developed flow assumption is validated
 - b. Numerical model with UDF for varying viscosities is validated, with special care taken in providing good guess for back flow temperatures
- Lower shell temperatures will generate higher velocity profiles, thus higher flux flow rate in the gap;
- Larger gap size tends to create more upward flows, thus less flux flow rate going out of the mold bottom;
- 4) Linear pressure distribution along the casting direction at the shell-flux interface might be a good guess in this problem, however, in cases with non-uniform temperature distribution the pressure will not linearly increase along the casting direction. And the fully-developed flow assumption will not be appropriate. In such cases, the numerical model should be used for predictions.

A.3. Scenario II: Liquid Flux Flow between Parallel Plates with Temperature-Dependent Viscosity

Liquid flux flow in the mold-shell gap is strongly affected by the heat transfer process in the gap, since the viscosity of the liquid flux is sensitive to the local temperature. The temperature distribution is influenced by the velocity field in the gap. An accurate prediction of the liquid flux velocity profile between the gap and corresponding consumption rate requires solving for the fully coupled momentum equations and the energy equation. However, simplifications of the original governing equations can be made so that an analytical solution can be sought.

The sketch for the problem is shown in Figure A.5, with the solidified flux layer and liquid flux layer between the mold wall and the solidified steel shell. The shell temperature is changing with the distance below the meniscus to approximate the real situation. The pressure distribution at the shell-molten steel interface is assumed linear, and the pressure distribution at the shell-molten steel interface is assumed linear, and the pressure distribution at the shell-liquid flux interface is not necessarily linear. Case I demonstrated that if the shell temperature is not uniform, then the fully-developed flow assumption will not work since the consumption rate will change at different locations along the casting direction. This is not consistent with the mass conservation principle. Thus in the real situation, the flow is not fully developed, and the pressure at shell-liquid flux interface is not linearly distributed, in order to satisfy the requirement for mass conservation. Although the fully-developed flow assumption is not used to simplify the governing equations, instead the scaling technique will be used for the validation of the assumptions used in the solution process.

The assumptions for this scenario are listed below:

1) Newtonian, incompressible fluid for the liquid flux.

- 2) Steady state reached for both fluid flow and heat transfer/solidification in the gap.
- 3) Real curvatures of the mold and shell are not considered in this analysis, and the liquid flux layer is assumed to have a fixed thickness between parallel plates.
- 4) The left plate approximating the solid-liquid flux interface has a constant temperature, which is the solidification temperature of the flux, and the right plate has the shell temperature varying along the casting direction
- 5) Vertical velocity change along the casting direction is negligible.
- 6) Horizontal velocity (transverse velocity) between the gap is negligible.
- 7) Heat conduction along the casting direction is ignored.
- Pressure is constant across the gap, but varying along the casting direction. This is true for a thin liquid flux layer as is the case here.



Figure A.5. Sketch for the domain of analysis

The simplified governing equations based on the above assumptions for this process include: Continuity equation:

$$\frac{\partial V_x}{\partial x} + \frac{\partial V_z}{\partial z} = 0 \tag{A.21}$$

and further $V_x = 0$ based on the impermeable wall boundary condition, and the assumption that Z-momentum equation:

$$0 = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \frac{1}{\rho} \left(\frac{\partial}{\partial x} \left(\mu \frac{\partial V_z}{\partial x} \right) \right) + g \text{, or } \frac{d}{dx} \left(\mu \frac{dV_z}{dx} \right) = \frac{dP}{dz} - \rho g \tag{A.22}$$

Energy equation:

$$0 = k \left(\frac{\partial^2 T}{\partial x^2}\right), \text{ or } \frac{\partial^2 T}{\partial x^2} = 0$$
(A.23)

So the two equations (A.22) and (A.23) to solve include the simplified Z-direction momentum equation and the simplified energy equation with the boundary conditions imposed for each of the equations. The pressure gradient along the casting direction is currently unknown.

The simplified energy equation is simply a 1-D diffusion equation. Ignoring the advection term in the energy equation will decrease the accuracy of the temperature distribution; however, this simplification significantly reduces the complexity of the analysis. Thus this 1-D diffusion equation is solvable. Further improvement will be made if necessary after comparing this solution to the numerical model which is employed to evaluation the analytical model and the assumptions. Owing to the varying shell temperature along the casting direction, the boundary condition is made a function of the casting direction. Thus the solution is:

$$T(x,z) = T_{fsol} + \frac{T_{shell}(z) - T_{fsol}}{H_{liq}}x$$
(A.24)

Integrating equation (A.22) results in the form below:

$$\mu \frac{dV_z}{dx} = \left(\frac{dp}{dz} - \rho_{slag}g\right)x + C_0 \tag{A.25}$$

Second integration of the equation above leads to:

$$V_z = \int \frac{1}{\mu} \left(\left(\frac{dp}{dz} - \rho_{slag} g \right) x + C_0 \right) dx + C_1$$
(A.26)

Further evaluation of the integration above needs the temperature distribution in the domain. Thus the temperature profile is plugged into the velocity expression above for further derivation.

$$\mu = \mu_0 \left(\frac{T_0 - T_{fsol}}{T - T_{fsol}}\right)^n = \mu_0 \left(\frac{T_0 - T_{fsol}}{T_{fsol} + \frac{T_{shell}(z) - T_{fsol}}{H_{liq}}x - T_{fsol}}\right)^n = \mu_0 \left(\frac{T_0 - T_{fsol}}{\frac{T_0 - T_{fsol}}{H_{liq}}x}\right)^n$$
(A.27)

And

$$V_{z} = \frac{1}{\mu_{0} \left(T_{0} - T_{fsol}\right)^{n}} \left(\frac{T_{shell}(z) - T_{fsol}}{H_{liq}}\right)^{n} \left(\int \left(\frac{dp}{dz} - \rho_{slag}g\right) x^{n+1} dx + \int C_{0} x^{n} dx\right) + C_{1}$$
(A.28)

The final form for the velocity solution follows

$$V_{z} = \frac{1}{\mu_{0}H_{liq}^{n}} \left(\frac{T_{shell}(z) - T_{fsol}}{T_{0} - T_{fsol}}\right)^{n} \left(\left(\frac{dp}{dz} - \rho_{slag}g\right)\frac{x^{n+2}}{n+2} + \frac{C_{0}x^{n+1}}{n+1}\right) + C_{1}$$
(A.29)

The integration constants are determined by the boundary condition for the velocity.

At x = 0, the velocity is zero, due to the no-slip wall B.C., resulting in $C_1 = 0$.

At $x = H_{liq}$, the velocity is the casting speed, due to the no-slip wall B.C., leading to

$$C_{0} = V_{casting} \frac{\mu_{0} (n+1) (T_{0} - T_{fsol})^{n}}{H_{liq} (T_{shell} (z) - T_{fsol})^{n}} - \left(\frac{dp}{dz} - \rho_{slag} g\right) \frac{n+1}{n+2} H_{liq}$$
(A.30)

So the solution to the Z-momentum equation is written as

$$V_{z} = \frac{1}{\mu_{0}H_{liq}^{n}} \left(\frac{T_{shell}(z) - T_{fsol}}{T_{0} - T_{fsol}}\right)^{n} \left(\left(\frac{dp}{dz} - \rho_{slag}g\right)\frac{x^{n+2}}{n+2} + \frac{C_{0}x^{n+1}}{n+1}\right)$$
(A.31)

With

$$C_{0} = V_{casting} \frac{\mu_{0} (n+1) (T_{0} - T_{fsol})^{n}}{H_{liq} (T_{shell} (z) - T_{fsol})^{n}} - \left(\frac{dp}{dz} - \rho_{slag} g\right) \frac{n+1}{n+2} H_{liq}$$
(A.32)

The pressure gradient along the casting direction is not known, so the next step in the derivation will focus on investigating the pressure gradient via looking at the overall mass conservation in the domain. The volume flow rate of the liquid flux is expressed by equation (A.33), with C_0 shown in equation (A.32):

$$Q = \rho W \int_{0}^{H_{liq}} V_z dx = \frac{\rho W}{\mu_0 H_{liq}} \int_{0}^{H_{liq}} \left(\frac{T_{shell}(z) - T_{fsol}}{T_0 - T_{fsol}} \right)^n \left(\left(\frac{dp}{dz} - \rho_{slag} g \right) \frac{x^{n+2}}{n+2} + \frac{C_0 x^{n+1}}{n+1} \right) dx$$
(A.33)

Further simplification of the expression above leads to

$$Q = \frac{\rho W}{\mu_0 H_{liq}^{n}} \left(\frac{T_{shell}(z) - T_{fsol}}{T_0 - T_{fsol}} \right)^n \left(\left(\frac{dp}{dz} - \rho_{slag} g \right) \frac{H_{liq}^{n+3}}{(n+2)(n+3)} + \frac{C_0 H_{liq}^{n+2}}{(n+1)(n+2)} \right)$$
(A.34)

Inserting the expression for C_0 into the equation above,

$$Q = \frac{\rho W}{\mu_0} \left(\frac{T_{shell}(z) - T_{fsol}}{T_0 - T_{fsol}} \right)^n \left(\frac{\left(\frac{dp}{dz} - \rho_{slag} g \right) \frac{H_{liq}^3}{(n+2)(n+3)} + \frac{H_{liq}^2}{(n+1)(n+2)} \left(V_{casting} \frac{\mu_0 (n+1) \left(T_0 - T_{fsol} \right)^n}{H_{liq} \left(T_{shell}(z) - T_{fsol} \right)^n} - \left(\frac{dp}{dz} - \rho_{slag} g \right) \frac{n+1}{n+2} H_{liq} \right) \right)$$

(A.35)

The final form of the equation above follows

$$Q = \frac{\rho W V_{casting} H_{liq}}{(n+2)} - \frac{\rho W H_{liq}^3}{\mu_0 (n+2)^2 (n+3)} \left(\frac{T_{shell} (z) - T_{fsol}}{T_0 - T_{fsol}}\right)^n \left(\frac{dp}{dz} - \rho_{slag} g\right)$$
(A.36)

The law of mass conservation requires a constant volume flow rate of the liquid flux in the gap. In order to force the flow rate to be constant after the assumptions used in the simplification of the original model, the gradient of this flow rate along the casting direction should be zero.

$$\frac{dQ}{dz} = 0 \tag{A.37}$$

Combining equation (12) and (13), the equation for pressure gradient is derived as:

$$\frac{d}{dz}\left(\left(T_{shell}\left(z\right)-T_{fsol}\right)^{n}\left(\frac{dp}{dz}-\rho_{slag}g\right)\right)=0$$
(A.38)

Then the ODE for pressure is derived from equation (14) as

$$n\frac{dT_{shell}(z)}{dz}\left(\frac{dp}{dz}-\rho_{slag}g\right)+\frac{d^2p}{dz^2}\left(T_{shell}(z)-T_{fsol}\right)=0$$
(A.39)

Let $\psi = \frac{dp}{dz}$,

$$\frac{d\psi}{dT_{shell}(z)} = -n \frac{\psi - \rho_{slag}g}{T_{shell}(z) - T_{fsol}}$$
(A.40)

and

$$\int \frac{d\psi}{\psi - \rho_{slag}g} = -n \int \frac{dT_{shell}(z)}{T_{shell}(z) - T_{fsol}} + C_3$$
(A.41)

Integration leads to

$$\ln\left(\psi - \rho_{slag} g\right) = \ln\left[C_4 \left(T_{shell} \left(z\right) - T_{fsol}\right)^{-n}\right]$$
(A.42)

and the solution is

$$\psi = \rho_{slag}g + C_4 \left(T_{shell}(z) - T_{fsol}\right)^{-n} \tag{A.43}$$

So the pressure gradient along the casting direction is written as

$$\frac{dp}{dz} = \rho_{slag}g + C_4 \left(T_{shell}(z) - T_{fsol}\right)^{-n}$$
(A.44)

And the pressure distribution is solved as

$$p = \rho_{slag} gz + C_4 \int \left(T_{shell} \left(z \right) - T_{fsol} \right)^{-n} dz + C_5$$
 (A.45)

The integration constants in the equation above are determined by the pressure values at the upper inlet and the lower outlet (serving as boundary conditions).

At z = 0, the pressure can be approximated as 1 atm.

At z = L, the pressure takes the ferro-static pressure on the other side of the shell. (At z=L the pressure is atmosphere, but if we use this as the boundary condition, information about the pressure will be lost. Therefore we use the condition for $z=\sim L$ to be the condition for z=L.)

The integration of the temperature along the casting direction requires the temperature be a function of the distance in casting direction. This function could be a curve fit from the measurement or previous numerical results. According to the data from CON1D calculation, the temperature at shell-liquid flux interface is approximately varying linearly down the shell if the entrance effect is ignored. So currently the temperature distribution at the shell-liquid flux interface is assumed a linear function of the distance down the mold, shown in Fig. 2. The real temperature profile is use in Case III.



Figure A.6. Shell temperature (linear variation)

Assume at the gap inlet (z = 0), the temperature is fixed at $T_{in} = 1250$ C, and at $T_{out} = 1160$ C at the gap outlet (z = 0.8 m). Then the temperature distribution writes

$$T_{shell}(z) = T_{in} + \frac{T_{out} - T_{in}}{L} z \quad (0 < z < L)$$
(A.46)

Resulting in

$$p = \rho_{slag} gz + C_4 \frac{L}{(T_{out} - T_{in})(1 - n)} \left(\frac{T_{out} - T_{in}}{L} z + T_{in} - T_{fsol}\right)^{1 - n} + C_5$$
(A.47)

According to the pressure boundary values discussed above,

$$C_4 \frac{L}{(T_{out} - T_{in})(1 - n)} (T_{in} - T_{fsol})^{1 - n} + C_5 = p_{in}$$
(A.48)

$$C_{4} \frac{L}{(T_{out} - T_{in})(1 - n)} (T_{out} - T_{fsol})^{1 - n} + C_{5} = p_{out} - \rho_{slag} gL$$
(A.49)

Solving for the constants in the two equations (A.48) and (A.49) above, resulting in the form:

$$C_{4} = \frac{1}{L} \frac{\left(p_{out} - p_{in} - \rho_{slag} gL\right) \left(T_{out} - T_{in}\right) \left(1 - n\right)}{\left(T_{out} - T_{fsol}\right)^{1 - n} - \left(T_{in} - T_{fsol}\right)^{1 - n}}$$
(A.50)

$$C_{5} = p_{in} - \frac{\left(p_{out} - p_{in} - \rho_{slag}gL\right)\left(T_{in} - T_{fsol}\right)^{1-n}}{\left(T_{out} - T_{fsol}\right)^{1-n} - \left(T_{in} - T_{fsol}\right)^{1-n}}$$

$$= \frac{p_{in}\left(T_{out} - T_{fsol}\right)^{1-n} - p_{out}\left(T_{in} - T_{fsol}\right)^{1-n} + \rho_{slag}gL\left(T_{in} - T_{fsol}\right)^{1-n}}{\left(T_{out} - T_{fsol}\right)^{1-n} - \left(T_{in} - T_{fsol}\right)^{1-n}}$$
(A.51)

Thus the final form for the pressure distribution is shown in equation (A.52):

$$p = \rho_{slag} gz + \frac{\left(p_{out} - p_{in} - \rho_{slag} gL\right)}{\left(T_{out} - T_{fsol}\right)^{1-n} - \left(T_{in} - T_{fsol}\right)^{1-n}} \left(\frac{T_{out} - T_{in}}{L} z + T_{in} - T_{fsol}\right)^{1-n}} + \frac{p_{in} \left(T_{out} - T_{fsol}\right)^{1-n} - p_{out} \left(T_{in} - T_{fsol}\right)^{1-n} + \rho_{slag} gL \left(T_{in} - T_{fsol}\right)^{1-n}}{\left(T_{out} - T_{fsol}\right)^{1-n} - \left(T_{in} - T_{fsol}\right)^{1-n}}$$
(A.52)

And the pressure gradient is expressed as:

$$\frac{dp}{dz} = \rho_{slag}g + \frac{1}{L} \frac{\left(p_{out} - p_{in} - \rho_{slag}gL\right)\left(T_{out} - T_{in}\right)\left(1 - n\right)}{\left(T_{out} - T_{fsol}\right)^{1 - n} - \left(T_{in} - T_{fsol}\right)^{1 - n}} \left(T_{shell}\left(z\right) - T_{fsol}\right)^{-n}$$
(A.53)

With this formula for the pressure distribution, the downward velocity profile across the gap and the volume flow rate are finalized as:

$$V_{z} = \frac{1}{\mu_{0}H_{liq}^{n}} \left(\frac{T_{shell}(z) - T_{fsol}}{T_{0} - T_{fsol}}\right)^{n} \left(\left(\frac{1}{L} \frac{\left(p_{out} - p_{in} - \rho_{slag}gL\right)\left(T_{out} - T_{in}\right)\left(1 - n\right)}{\left(T_{out} - T_{fsol}\right)^{1 - n} - \left(T_{in} - T_{fsol}\right)^{1 - n}} \left(T_{shell}(z) - T_{fsol}\right)^{-n}\right) \frac{x^{n+2}}{n+2} + \frac{C_{0}x^{n+1}}{n+1}\right)$$
(A.54)

which further simplifies into

$$V_{z} = \frac{V_{casting}}{H_{liq}^{n+1}} x^{n+1} + \frac{1}{L(n+2)\mu_{0}H_{liq}^{n}} \left(\frac{1}{T_{0} - T_{fsol}}\right)^{n} \left(\frac{\left(p_{out} - p_{in} - \rho_{slag}gL\right)\left(T_{out} - T_{in}\right)\left(1 - n\right)}{\left(T_{out} - T_{fsol}\right)^{1 - n} - \left(T_{in} - T_{fsol}\right)^{1 - n}}\right) \left(x^{n+2} - x^{n+1}H_{liq}\right)$$
(A.55)

$$Q = \frac{\rho W V_{casting} H_{liq}}{(n+2)} - \frac{\rho W H_{liq}^3}{L \mu_0 (n+2)^2 (n+3) (T_0 - T_{fsol})^n} \frac{(p_{out} - p_{in} - \rho_{slag} g L) (T_{out} - T_{in}) (1-n)}{(T_{out} - T_{fsol})^{1-n} - (T_{in} - T_{fsol})^{1-n}}$$
(A.56)

From the equation of the volume flow rate (or consumption rate), the flow rate is not changing for any slice along the casting direction, so mass is conserved in this system.

According to the analytical solutions derived above, the velocity profile across the liquid flux layer is obtained, as well as the temperature distribution. In current case, the temperature is linearly changing from the top to the bottom of the mold in the liquid flux layer. The other parameters used in this case are listed in the problem statement. Pressure distribution is plotted in Figure A.7, and the vertical velocity profile is shown in Figure A.8 below.



Figure A.7. Pressure distribution down the mold (Scenario I linear distribution vs. calculated)



Figure A.8. Pressure gradient down the mold (Scenario I linear distribution vs. calculated)

The velocity profile across the liquid flux layer is shown in Figure A.9.



Figure A.9. Downward velocity profile across the gap (Scenario II)

Figure A.10 shows the comparison of velocity profiles from current model with those from Case II, where the shell temperature is constant.



Figure A.10. Comparison of velocity profiles from Scenario I and current model

From the derivation of the model and the results, it is shown that the pressure is the key factor to enforce the mass conservation in the system. In Case I, in which case the shell temperature is fixed, the linear pressure distribution will not necessarily satisfy the mass balance in the system. In the current model, the pressure is calculated as a function of the temperature distribution at the shell. So the varying temperature effect has been taken into account.

In this case, the temperature is linearly distributed just for simplicity. Equations (20) and (21), derived for the pressure and its gradient respectively, are suitable for solving with any temperature distribution.

A.4. Scenario III: Liquid Flux Flow between Parallel Plates with Varying Gap Size and Temperature-Dependent Viscosity

The analysis presented in Scenario II can be generalized to incorporate any gap and temperature profile. In Scenario II, the relationship between the pressure (gradient) distribution and the velocity profile in the liquid gap were derived as in equations (A.31), (A.32) and (A.36). Based on mass conservation, the flow rate of the liquid flux follows equation (A.37).

In this scenario, the liquid flux thickness, H_{lig} , is also a function of the distance below meniscus (z-direction distance down along the shell). So the differentiation of equation (A.3) leads to:

$$V_{casting} \frac{dH_{liq}}{dz} - \frac{1}{\mu_0 (n+2)(n+3)(T_0 - T_{fsol})^n} \begin{pmatrix} 3H_{liq}^2 \frac{dH_{liq}}{dz} (T_{shell}(z) - T_{fsol})^n \left(\frac{dp}{dz} - \rho_{slag}g\right) \\ + nH_{liq}^3 (T_{shell}(z) - T_{fsol})^{n-1} \frac{dT_{shell}}{dz} \left(\frac{dp}{dz} - \rho_{slag}g\right) \\ + H_{liq}^3 (T_{shell}(z) - T_{fsol})^n \frac{d^2p}{dz^2} \end{pmatrix} = 0$$
(A.57)

The shell temperature is changing in the casting direction with the following curve-fitted polynomial functions that are used in the numerical study to approximate the real case.



Figure A.11. Temperature distribution along casting direction

The corresponding liquid layer thickness also changes along the casting direction, following:

Narrow face:
$$H(z) = H_S - H_M - H_T$$

 $H_T = 1.3 + (1.0\%/m)^* (W/2)^* z/L = 1.3 + 0.008125z$
 $H_M = 4Y \left(\frac{z}{L} - \frac{z^2}{L^2}\right)$
 $H(z) = 2 + 0.02z - 0.000012z^2 - 4 \left(\frac{z}{L} - \frac{z^2}{L^2}\right) - 1.3 - 0.008125z$
 $H(z) = -0.00000575z^2 + 0.006875z + 0.7$
Wide face: $H(z) = 0.95 + 1.75z/L = 0.95 + 0.0021875z$
 $H(z) = 0.95 + 0.002187z$
Corner: $H_S(z) = 2 + 0.02z - 0.000012z^2$

Reorganization of the equations (5) above gives:

$$\frac{d^{2}p}{dz^{2}} + \left(\frac{3}{H_{liq}}\frac{dH_{liq}}{dz} + \frac{n}{(T_{shell} - T_{fsol})}\frac{dT_{shell}}{dz}\right)\frac{dp}{dz} = \frac{V_{casting}}{H_{liq}^{3}}\frac{dH_{liq}}{dz}\frac{\mu_{0}(n+2)(n+3)(T_{0} - T_{fsol})^{n}}{(T_{shell} - T_{fsol})^{n}} + \left(\frac{3}{H_{liq}}\frac{dH_{liq}}{dz} + \frac{n}{(T_{shell} - T_{fsol})}\frac{dT_{shell}}{dz}\right)\rho g$$
(A.58)

This ODE contains only the second and first derivatives of pressure, so a simple substitution of the pressure gradient leads to

$$\frac{dp'}{dz} + \left(\frac{3}{H_{liq}}\frac{dH_{liq}}{dz} + \frac{n}{(T_{shell} - T_{fsol})}\frac{dT_{shell}}{dz}\right)p' = \frac{V_{casting}}{H_{liq}^3}\frac{dH_{liq}}{dz}\frac{\mu_0(n+2)(n+3)(T_0 - T_{fsol})^n}{(T_{shell} - T_{fsol})^n} + \left(\frac{3}{H_{liq}}\frac{dH_{liq}}{dz} + \frac{n}{(T_{shell} - T_{fsol})}\frac{dT_{shell}}{dz}\right)\rho g$$
(A.59)

where $p' = \frac{dp}{dz}$

The simpler form of the equation above is

$$\frac{dp'}{dz} + f_1(z)p' = f_0(z) + f_1(z)\rho g$$
(A.60)

where

$$f_1(z) = \left(\frac{3}{H_{liq}}\frac{dH_{liq}}{dz} + \frac{n}{(T_{shell} - T_{fsol})}\frac{dT_{shell}}{dz}\right)$$
(A.61)

$$f_0(z) = \frac{V_{casting}}{H_{liq}^3} \frac{dH_{liq}}{dz} \frac{\mu_0 (n+2)(n+3)(T_0 - T_{fsol})^n}{(T_{shell} - T_{fsol})^n}$$
(A.62)

Solution to the ODE above has a general form, which follows the solution to the first-order linear differential equation:

$$p' = Ce^{\int -f_1(z)dz} + e^{\int -f_1(z)dz} \int e^{\int f_1(z)dz} (f_0(z) + f_1(z)\rho g) dz$$
(A.63)

After simplifying the results, the solution to the equation above writes:

$$p' = \frac{C}{H_{liq}^3 (T_{shell} - T_{fsol})^n} + \frac{V_{casting} \mu_0 (n+2)(n+3)(T_0 - T_{fsol})^n}{H_{liq}^2 (T_{shell} - T_{fsol})^n} + \rho g$$
(A.64)

The direct integration of the equation above leads to too complicated algebraic derivations, so the numerical integration of the pressure gradient in equation (11) will help to decide the constants, with the fixed pressure values at the gap inlet and outlet.

Then the pressure distribution can be achieved as:

$$p = \int p' dz + C_1 \tag{A.65}$$

where parameter C is determined via the numerical integration on equation (A.64), which will further be used for the velocity and flow rate equations.

Integration of the equation (A.66) results in:

The integration is divided into 3 parts, shown in equation (13). The pressure distribution can be written into the following form:

$$p(z) = C * II(z) + I2(z) + I3(z) + C_1$$
(A.67)

Two sets of boundary conditions are used in current study to determine the constants C and C_1 . The numerical integrations are performed using Matlab, and the two constants are solved and shown in Table 2.

1. Boundary Condition Type 1:

At *z*=1 mm below meniscus, the pressure is **1 atm**;

At z=800 mm below meniscus, pressure takes the **ferrostatic pressure** at 0.8 m below meniscus.

2. Boundary Condition Type 2:

At *z*=1 mm below meniscus, the pressure is **1** *atm*;

At z=800 mm below meniscus, the pressure takes 1 atm at 0.8 m below meniscus.

Table II.	Numerical Integration	n Parameters and Consta	ants
	WF	NF	CORNER
I1(1)	10.3563	79.6154	5.8120
I2(1)	0.3382	1.9157	0.3996
I3(1)	0.2629	0.2629	0.2629
I1(NX)	4.3079e6	3.5357e+005	1.8606e+004
I2(NX)	2.3403e5	1.5329e+004	1.9284e+003
I3(NX)	2.1033e4	2.1033e+004	2.1033e+004
C (B.C. type 1)	-0.0459	0.0599	1.8584
C (B.C. type 2)	-0.0592	-0.1029	-1.2344

- - - --NT · 1 T . -10

$$V_{z} = \frac{1}{\mu_{0}H_{liq}^{n}} \left(\frac{T_{shell}(z) - T_{fsol}}{T_{0} - T_{fsol}}\right)^{n} \left(\left(\frac{dp}{dz} - \rho_{slag}g\right)\frac{x^{n+2}}{n+2} + \frac{C_{0}x^{n+1}}{n+1}\right)$$
(A.68)

$$C_{0} = V_{casting} \frac{\mu_{0} (n+1) (T_{0} - T_{fsol})^{n}}{H_{liq} (T_{shell} (z) - T_{fsol})^{n}} - \left(\frac{dp}{dz} - \rho_{slag} g\right) \frac{n+1}{n+2} H_{liq}$$
(A.69)

$$\frac{dp}{dz} = \frac{C}{H_{liq}^3 (T_{shell} - T_{fsol})^n} + \frac{V_{casting} \mu_0 (n+2)(n+3)(T_0 - T_{fsol})^n}{H_{liq}^2 (T_{shell} - T_{fsol})^n} + \rho_{slag} g$$
(A.70)

Combining all three equations above, the velocity profile is finalized as:

$$V_{z} = V_{casting} \frac{x^{n+1}}{H_{liq}^{n+1}} + \left(\frac{C}{\mu_{0} H_{liq}^{n+3} (T_{0} - T_{fsol})^{n}} + \frac{V_{casting} (n+2)(n+3)}{H_{liq}^{n+2}}\right) \frac{x^{n+2} - x^{n+1} H_{liq}}{n+2}$$
(A.71)

For wide faces:



Figure A.12. Liquid flux layer thickness vs. distance down the mold (wide face)



Figure A.13. Comparison of pressure from different boundary conditions

So the liquid flux velocity profile in the wide face gap is solved as:

B.C. TYPE 1: (ferrostatic pressure at mold exit)



Figure A.14. Vertical velocity profile for B.C. type 1 (ferrostatic pressure at gap outlet)

B.C. TYPE 2: (atmospheric pressure at mold exit)



Figure A.15. Vertical velocity profile for B.C. type 2 (1 atm at gap outlet)

For narrow faces:



Figure A.16. Liquid flux layer thickness vs. distance down the mold (narrow face)



Figure A.17. Comparison of pressure from different boundary conditions

B.C. TYPE 1: (ferrostatic pressure at mold exit)



Figure A.18. Vertical velocity profile for B.C. type 1 (ferrostatic pressure at gap outlet)

B.C. TYPE 2: (atmospheric pressure at mold exit)



Figure A.19. Velocity profile across liquid flux layer (1 atm at the gap outlet at mold bottom)





Figure A.20. Liquid flux layer thickness vs. distance down the mold (mold corner)



Figure A.21. Comparison of pressure profile from different boundary conditions

B.C. TYPE 1: (ferrostatic pressure at mold exit)



Figure A.22. Downward velocity profile across the liquid flux layer (B.C. type 1)

B.C. TYPE 2: (atmospheric pressure at mold exit)



Figure A.23. Downward velocity profile across the liquid flux layer (B.C. type 2) Comparison of vertical velocity profiles at wide face gap, narrow face gap and the corner gap is shown bellow.



Figure A.24. Pressure distribution for different gaps



Figure A.25. Vertical velocity profile for different gaps at mold exit

Observations from figures above show that the boundary condition with pressure value of 1 atm at the mold exit is the correct pressure boundary value which can be used to calculate the velocity profile of the liquid flux. Since the gap is filled with liquid flux (and some solid flux), the shape of the gaps is determined by the shell deformation. Also the shell deformation is determined by the ferrostatic pressure of the molten steel. The shell itself has the strength to hold the liquid steel in the mold, and it deforms to form the gap. Thus the shell will not exert the ferrostatic pressure on the liquid flux. So the 1 atm pressure should be imposed on both the gap inlet and outlet as the boundary conditions.

As the gap size increases, the velocity of the liquid flux keeps increasing, and the flow rate also keeps increasing. For the corner gap, the velocity profile shows a peak about 2 mm from the shell, which suggests that the gravity is accelerating the bulk fluid where the fluid cannot "feel" the existence of the solid wall. However, for smaller liquid flux thickness, as in the narrow face gap and the wide face gap, the peak will not exist simply because of the high viscosity throughout the liquid layer.

A.5. Scaling and Validation for Model Assumptions

Scenario 1 – Assumption 4

The fully developed assumption can be justified by calculating the entrance length. This is calculated with the minimum mu. For higher viscosities, the entrance length will be shorter.

Entrance _ Length = .06 Re =
$$\frac{\rho V_{cast} H_{liq}}{\mu} = \frac{2680 * .025 * .001}{.321} = .2m$$

Scenario II – Continuity Scaling

Continuity equation:

$$\frac{\partial V_x}{\partial x} + \frac{\partial V_z}{\partial z} = 0 \tag{A.1}$$

By using informal scaling we can estimate the unknown velocity, V_x .

$$\frac{V_x}{H_{liq}} + \frac{V_z}{L} = 0$$
 therefore $V_x = V_{cast} \frac{H_{liq}}{L}$ which is $V_x = V_{cast} \frac{1}{800}$ and therefore it is negligible.

Scenario II – Z-momentum Scaling

Z-momentum equation:

$$\frac{\partial V_z}{\partial t} + V_x \frac{\partial V_z}{\partial x} + V_z \frac{\partial V_z}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \frac{1}{\rho} \left(\frac{\partial}{\partial x} \left(\mu \frac{\partial V_z}{\partial x} \right) + \frac{\partial}{\partial z} \left(\mu \frac{\partial V_z}{\partial z} \right) \right) + g$$
(A.3)

We apply the steady state condition and continuity scaling to reduce this equation.

$$V_{z}\frac{\partial V_{z}}{\partial z} = -\frac{1}{\rho}\frac{\partial P}{\partial z} + \frac{1}{\rho}\left(\frac{\partial}{\partial x}\left(\mu\frac{\partial V_{z}}{\partial x}\right) + \frac{\partial}{\partial z}\left(\mu\frac{\partial V_{z}}{\partial z}\right)\right) + g$$
(A.72)

By applying informal scaling and rearranging:

$$\frac{\Delta P}{L} - \rho g \sim \rho V_z \frac{V_z}{L} + \frac{1}{H_{liq}} \frac{\mu V_z}{H_{liq}} + \frac{\mu V_z}{L^2}$$
(A.73)

By recognizing the $\frac{1}{H_{liq}^2} >> \frac{1}{L^2}$ we can eliminate the Z viscosity term.

$$\frac{\Delta P}{L} - \rho g \sim \rho \frac{V_z^2}{L} + \frac{\mu V_z}{H_{liq}^2}$$
(A.74)

Comparing the advection and viscous term:

$$\rho \frac{V_z H_{liq}}{\mu} \frac{H_{liq}}{L} \to \operatorname{Re}^* \frac{1}{800}$$
(A.75)

Therefore the viscous term dominates and we can neglect the advection components.

Case II – Energy Scaling

Energy Equation:

$$\rho C_{P} \left(\frac{\partial T}{\partial t} + V_{x} \frac{\partial T}{\partial x} + V_{z} \frac{\partial T}{\partial z} \right) = k \left(\frac{\partial^{2} T}{\partial x^{2}} + \frac{\partial^{2} T}{\partial z^{2}} \right)$$
(A.4)

We apply the steady state condition and continuity scaling to reduce this equation.

$$\rho C_{P} \left(V_{z} \frac{\partial T}{\partial z} \right) = k \left(\frac{\partial^{2} T}{\partial x^{2}} + \frac{\partial^{2} T}{\partial z^{2}} \right)$$
(A.76)

By applying informal scaling:

$$\rho C_{P} \left(V_{z} \frac{\Delta T}{L} \right) \sim k \left(\frac{\Delta T}{H_{liq}^{2}} + \frac{\Delta T}{L^{2}} \right)$$
(A.77)

Again recognizing $\frac{1}{H_{liq}^2} >> \frac{1}{L^2}$ we can neglect the Z-term

$$\rho C_P \left(V_z \frac{\Delta T}{L} \right) \sim k \left(\frac{\Delta T}{H_{liq}^2} \right)$$
(A.78)

Comparing the advection to the conduction term:

$$\left(\frac{V_z H_{liq}}{L}\right) \frac{1}{L} \to Pe^* \frac{1}{800} \tag{A.79}$$

Therefore the conduction term dominates and the advection terms can be neglected.

APPENDIX E. PUBLICATIONS DURING PH.D STUDY

Papers published during Ph.D study include:

	[1] [2]	 R. Liu, et al., Slidegate Dithering Effects on Transient Flow and Mold Level Fluctuations. Iron & Steel Technology, 2014, in press. R. Liu, S.P. Vanka and B.G. Thomas, Particle Transport and Deposition in a Turbulent Square Duct Flow with an Imposed Magnetic Field. Journal of Fluids Engineering, 2014, in press.
Journal Papers	[3]	R. Liu et al, Measurements of Molten Steel Surface Velocity and Effects of Stopper-rod Movements on Transient Multiphase Fluid Flow in Continuous Casting. (accepted by ISIJ International)
	[4]	R. Liu et al, Model of Gas Flow through Porous Refractory Applied to Upper Tundish Nozzles. (accepted by journal of Metallurgical and Materials Transactions B)
	[5]	R. Liu , J. Sengupta, M.M. Yavuz, and B.G. Thomas. Effects of Stopper Rod Movement on Mold Fluid Flow at ArcelorMittal Dofasco's No. 1 Continuous Caster. <i>AISTech 2011 proc.</i> , Indianapolis, IN, May 2-5, 2011, pp. 1619-1630
	[6]	R. Liu , J. Sengupta, D. Crosbie, S. Chung, M. Trinh and B.G. Thomas, Measurement of Molten Steel Surface Velocity with SVC and Nail Dipping during Continuous Casting Process. <i>TMS Annual Meeting</i> , San Diego, CA, 2011, pp.51-58
	[7]	R. Liu , B.G. Thomas and J. Sengupta, Simulation of Transient Fluid Flow in Mold Region during Steel Continuous Casting. <i>IOP Conf. Series: Materials Science and Engineering 33</i> , (MCWASP XIII conference, Austria, June, 2012), pp. 1-10 (peer reviewed)
	[8]	R. Liu and B.G. Thomas, Model of Gas Flow through Upper Tundish Nozzle Refractory and Initial Bubble Size. <i>Iron and Steel Technology Conference Proceedings</i> , (<i>AISTech conf.</i> , Atlanta, GA) 2012, pp. 2235-2245
First-author Conference Papers	[9]	R. Liu , B.G. Thomas, B. Forman and H. Yin, Transient Turbulent Flow Simulation with Water Model Validation and Application to Slide Gate Dithering, <i>Iron and Steel Technology Conference Proceedings</i> , (AISTech conf., Atlanta, GA), 2012, pp. 1317-1329
	[10]	R. Liu , B.G. Thomas, L. Kalra, T. Bhattacharya and A. Dasgupta, Slide-Gate Dithering Effects on Transient Flow and Mold Level Fluctuations. <i>Iron and Steel Technology Conference Proceedings</i> , (<i>AISTech conf.</i> , Pittsburgh, PA), 2013, pp. 1315-1328.
	[11]	R. Liu , S.P. Vanka and B.G. Thomas, Particle Transport and Deposition in a Turbulent Square Duct Flow with an Imposed Magnetic Field. <i>American Society of Mechanical Engineering (ASME) conf. proc.</i> , FEDSM2013-16503, Lake Tahoe, NV, July 09-11, 2013 (peer reviewed)
	[12]	R. Liu and S.P. Vanka, Eulerian-Lagrangian Simulation of Turbulent Upward Flows and Bubble Dynamics in a Vertical Square Duct, APS-DFD, Pittsburgh, 2013.
	[13]	B.G. Thomas, Q. Yuan, S. Mahmood, R. Liu , et al., Transport and Entrapment of Particles in Steel
Co-authored	[14]	S. Koric, L.C. Hibbeler, R. Liu , and B.G. Thomas, Multiphysics Model of Metal Solidification on
Journal Lapers	[15]	the Continuum Level. Numerical Heat Transfer, Part B: Fundamentals, 58(6), pp.371, 2010 L. Zhou, W. Wang, R. Liu , and B. G. Thomas. Computational Modeling of Temperature. Flow and
	[10]	Crystallization of Mold Slag during Double-Hot-Thermocouple-Technique Experiments. Metallurgical and Materials Transactions B, 44B, 2013, pp. 1264-1279.
Co-authored	[16] [17]	B.G. Thomas, R. Liu and B. Rietow, 8 th ECCC, (Austria), (2014) S. Koric, B. G. Thomas, and R. Liu , Proc. of the 8th Internat. Congress on Thermal Stresses,
Conference	[19]	TS2009, pp.249-253, (2009) B.G. Thomas S. Koric, I.C. Hibbeler and R. Liu , <i>STEELSIM2011</i> (Dusseldorf Germany) (2011)
Papers	[18] [19] [20]	B.G. Thomas, Q. Yuan, R. Liu , S. Mahmood, and R. Chaudhary, TMS, PA, pp.279, (2011) L.C. Hibbeler, R. Liu and B.G. Thomas, 7th ECCC, (Düsseldorf, Germany), (2011)
APPENDIX C. DERIVATION OF ANALYTICAL SOLUTIONS FOR 1-D PRESSURE-SOURCE MODEL IN CYLINDRICAL COORDINATE SYSTEM WITH CONSTANT GAS VISCOSITY

The one-way coupled heat conduction and pressure-source Equations 2 and 9 simplify into two ODEs, expressed in cylindrical coordinates as Equations A.5 and A.7. The heat conduction equation,

$$\frac{1}{r}(rT')' = 0 \tag{C.1}$$

has the general solution,

$$T = C_1 \ln r + C_2 \tag{C.2}$$

The two integration constants, C_1 and C_2 , are determined from the fixed temperature boundary conditions at the inner (T_1) and outer radius (T_2):

$$C_{1} = \frac{T_{2} - T_{1}}{\ln\left(\frac{R_{2}}{R_{1}}\right)} \text{ and } C_{2} = \frac{T_{1} \ln R_{2} - T_{2} \ln R_{1}}{\ln\left(\frac{R_{2}}{R_{1}}\right)}$$
(C.3)

The 1-D gas pressure equation can be expressed as:

$$p'' + \left(\frac{1}{r} - \frac{T'}{T} + \frac{K_D'}{K_D}\right)p' + \frac{p'^2}{p} = 0$$
(C.4)

Equation A.8 is solved for three different scenarios: 1) with both thermal expansion and temperature-dependent viscosity of the gas; 2) with gas thermal expansion and temperature-independent gas viscosity; and 3) without any thermal effects.

1. With thermal expansion and temperature-dependent gas viscosity

Starting with the most general scenario, the pressure Equation A.8 can be re-written as:

$$pp'' + p'^{2} + pp' \left(\frac{1}{r} - \frac{T'}{T} + \frac{K_{D}'}{K_{D}}\right) = 0$$
 (C.5)

Noting that $pp'' + p'^2 = (pp')'$, and $(pp') = \left(\frac{p^2}{2}\right)'$, Equation A.9 is further simplified into:

$$\frac{p^2}{2} = C_3 \int \frac{T}{rK_D} dr + C_4$$
 (C.6)

Using the relation $T'dT = C_1 \frac{dr}{r}$, Equation A.10 is converted into the following form:

$$\frac{p^2}{2} = \frac{C_3}{C_1 K_{DS}} \int T \mu(T) T' dT + C_4$$
(C.7)

Whether Equation A.10 can be solved analytically or not depends on the form of $\mu(T)$. For the empirical correlation for temperature-dependent gas viscosity adopted in the current work in Equation 19, there is no analytical solution for Equation A.11. Thus numerical solutions were obtained for this scenario by discretizing Equation A.9 using a central finite-difference scheme and solving the tri-diagonal matrix using the TDMA algorithm, on a 200-node mesh.^[39]

2. With thermal expansion and constant gas viscosity

With constant gas viscosity, the pressure Equation in A.9 simplifies into:

$$pp'' + p'^2 + pp'\left(\frac{1}{r} - \frac{T'}{T}\right) = 0$$
 (C.8)

Equation A.12 is further simplified into A.13 via a similar treatment from Equation A.9 to A.10

$$\frac{p^2}{2} = C_3 \int \frac{T}{r} dr + C_4 \tag{A.13}$$

Again using the relation $T'dT = C_1 \frac{dr}{r}$, equation (A.13) is converted into the following form:

$$\frac{p^2}{2} = \frac{C_3}{C_1} \int TT' dT + C_4 = \frac{C_3}{C_1} \frac{T^2}{2} + C_4 = \frac{C_3}{C_1} \frac{\left(C_1 \ln r + C_2\right)^2}{2} + C_4$$
(C.9)

Because pressure is always positive, Equation A.14 can be directly solved as:

$$p = \sqrt{\frac{C_3}{C_1} (C_1 \ln r + C_2)^2 + 2C_4}$$
(C.10)

The corresponding gas velocity is:

$$V_{r} = -\frac{K_{D}\sqrt{C_{1}C_{3}}\left(C_{1}\ln r + C_{2}\right)}{r\sqrt{\left(C_{1}\ln r + C_{2}\right)^{2} + \frac{2C_{1}C_{4}}{C_{3}}}}$$
(C.11)

where C_1 and C_2 are based on the temperature boundary conditions given in Equation A.7. Parameters C_3 and C_4 are determined in this scenario by the fixed pressure boundary conditions, given at the inner (P_1) and outer radius (P_2):

$$C_{3} = \frac{P_{2}^{2} - P_{1}^{2}}{T_{2}^{2} - T_{1}^{2}} C_{1} \text{ and } C_{4} = \frac{P_{1}^{2} T_{2}^{2} - P_{2}^{2} T_{1}^{2}}{2(T_{2}^{2} - T_{1}^{2})}$$
(C.12)

3. No thermal effects

The simplest scenario ignores both temperature-dependent gas viscosity and gas expansion. Assuming constant viscosity and temperature further simplifies Equation A.8 into the following diffusion equation:

$$p'' + \frac{p'}{r} = 0 (C.13)$$

The solution to this equation adopts the same form as the solution for the temperature distribution solved previously, and is written as:

$$p = C_3 \ln r + C_4 \tag{C.14}$$

where for this scenario,

$$C_{3} = \frac{P_{2} - P_{1}}{\ln \left(\frac{R_{2}}{R_{1}}\right)} \text{ and } C_{4} = \frac{P_{1} \ln R_{2} - P_{2} \ln R_{1}}{\ln \left(\frac{R_{2}}{R_{1}}\right)}$$
(C.15)

The corresponding gas velocity distribution is: $V_r = -K_D \frac{C_3}{r}$.

APPENDIX D.

ONE-WAY FLOW PRESSURE BOUNDARY CONDITION IN THE FORM OF ROBIN BOUNDARY CONDITION

The one-way-flow pressure boundary condition at refractory-liquid interface can be written in an efficient Robin-type B.C. form as Eq. (D.1).

$$\left(1 - a(\mathbf{x})\right)\frac{\partial p(\mathbf{x})}{\partial n} + a(\mathbf{x})p(\mathbf{x}) = a(\mathbf{x})g(\mathbf{x})$$
(D.1)

in which:

$$a(\mathbf{x}) = \frac{1 - sgn(\left(\frac{\partial p(\mathbf{x})}{\partial n}\right)_b)}{2}$$
(D.2)

$$sgn(x) = \begin{cases} -1, & if \quad x < 0, \\ 0, & if \quad x = 0, \\ 1, & if \quad x > 0. \end{cases}$$
(D.3)

$$g\left(\mathbf{x}\right) = p_{l}\left(\mathbf{x}\right) + \frac{2\sigma}{r_{pore}} \tag{D.4}$$

where the position vector \mathbf{x} moves along the refractory-metal interface (boundary).

APPENDIX E.

DERIVATION OF MASS AND MOMENTUM SINKS AT SHELL INTERFACE

Mass and momentum sinks were previously derived to account for the liquid steel solidification at shell interface.^[B1] The assumption used in the previous derivation was a planar shell profile, which is a reasonable approximation and simplifies the derivation. In this work, however, the mass and momentum terms are re-derived in a more general way taking into account the actual shape of the shell interface. Figure B.1 shows a schematic for the derivation with a curved shell interface shape. The shell surface divides the domain into liquid and solid regions, as shown in Figure B.1. This surface is stationary in the Eulerian frame of reference, but the solidified shell moves at a constant velocity, \mathbf{V}_{C} , in the casting direction.

Consider a Lagrangian shell surface element, **S**, that moves at the casting velocity, V_C . Within a time interval, Δt , this element moves from S at time t to a new position labeled S' in Figure B.1. This surface element has swept out a volume, ΔV , that contains the amount of steel that solidified during this time interval, Δt . This volume, ΔV , also equals the liquid steel mass loss across the shell surface element. Since liquid steel motion in this work is solved in an Eulerian reference frame with the finite volume approach, the control volume adjacent to this shell surface element should lose a mass of liquid steel, Δm , with volume ΔV , as given by Eq. (E.1), if the x-y plane is perpendicular to the casting direction:

$$\Delta V = \iint_{S_{xy}} \left[\left(z(x, y) + \Delta z \right) - z(x, y) \right] dxdy = \Delta z \iint_{S_{xy}} dxdy = \Delta z S_{xy} = V_C \Delta t S_{xy}$$
(E.1)

$$\Delta m = \rho \Delta V = V_C \Delta t S_{xy} \tag{E.2}$$

Generalizing to a non-vertical casting direction:

$$\Delta V = \Delta t \mathbf{V}_C \cdot \mathbf{S} \tag{E.3}$$

$$\Delta m = \rho \Delta t \mathbf{V}_c \cdot \mathbf{S} \tag{E.4}$$

where the dot product between the normal to the shell surface element, S, and a unit vector in the casting direction represents the projected area of the shell surface element, S_{xy} .

Shrinking the time step size, Eq. (E.2) can be re-written as a rate of mass loss as follows for the "mass sink":

$$\dot{m} = \rho \mathbf{V}_C \cdot \mathbf{S} = \rho V_C S_{xy} \tag{E.5}$$

The momentum sink is then computed from the results, knowing the local velocity field, V:

$$\dot{\mathbf{P}} = \dot{m}\mathbf{V} \tag{E.6}$$

These mass and momentum sinks just derived for an arbitrary-shaped concave shell interface, simplify to the formulations in previous work for planar interfaces.

This derivation in the current work reveals that the mass and momentum sinks do not depend on the actual shape of the shell surface (element), but depends only on the projected area of the shell surface (element) in the casting direction. Thus, Eqs. (E.3) and (E.4) comprise a more general formulation to account for the mass loss across the shell interface due to solidification in Eulerian mold flow simulations.



Figure E.1. Control volume adjacent to shell interface

Reference:

[E1] Q. Yuan, B.G. Thomas and S. P. Vanka, "Study of Transient Flow and Particle Transport during Continuous Casting of Steel Slabs, Part 1. Fluid Flow", *Metallurgical and Materials Transactions B*, 35B(4), 2004, pp. 685-702.