© by Bin Zhao. All rights reserved

NUMERICAL STUDY OF HEAT TRANSFER IN CONTINUOUS CASTING OF STEEL

BY

BIN ZHAO

B.E., Tsinghua University, 2000

THESIS

Submitted in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering in the Graduate College of the University of Illinois at Urbana-Champaign, 2003

Urbana, Illinois

Abstract

This thesis studies several problems related to the process of continuous casting of steel using numerical simulations. The problems investigated include steady state fluid flow and heat transfer in the the liquid flux layer, transient circular impinging jet heat transfer and transient fluid flow and heat transfer in the liquid steel pool in the mold region.

Numerical simulations are performed to study coupled fluid flow and heat transfer in a thin liquid slag or flux layer. The steady state Navier-Stokes equations are solved using the commercial finite volume code FLUENT. The combined effects of natural convection, bottom shear velocity and strongly temperature dependent viscosity are investigated. It is found that the variation of Nu with Ra for fluxes with strongly temperature dependent viscosities is analogous to correlations for fluids with constant viscosity, but the critical Ra number for the onset of natural convection is larger. For thin layers of realistic fluxes, natural convection is suppressed, and Nu increases linearly with increase of bottom shear velocity. The increase is greater for decreasing average viscosity. The increase of Nu is slight and is only due to end effects for the flat interface shape studied here.

Circular air jet impingement heat transfer is studied using large eddy simulations. Several simulations with different jet Reynolds numbers were carried out. The distance between jet exit and the impingement plate is 5 jet diameters. The simulation results are compared with experiments in the literature. It is found that the numerical model is capable of predicting the impinging jet heat transfer accurately. The simulations of impinging jet heat transfer also serve as validation cases for applying the numerical model to the more complicated problem of continuous casting.

The turbulent flow of molten steel and superheat in the mold region of a continuous caster of thin steel slabs is investigated with transient large eddy simulations and plant experiments. The computational model is validated through comparison with previous measurements of heat transfer during the impingement of an air jet on a cooled flat plate. The predicted fluid velocities match measurements taken from die injection experiments on full scale water models of the process. The corresponding predicted temperatures match measurements of thermocouples lowered into the molten steel during continuous casting. A classic double-roll flow pattern is confirmed for the 3-port nozzle and single phase flow of this operation. The results show that temperature in the top of the molten pool is 20-30% of the superheat temperature difference. Twelve percent of the superheat is extracted from the narrow face of this 132 mm-thick caster, where the peak heat flux reaches 750 kW/m^2 . 64% of the superheat is removed in the mold. The jets exiting the nozzle ports are shown to fluctuate, producing temperature fluctuations in the upper liquid pool of ± 4 °C and peak heat flux variations of $\pm 350 \ kW/m^2$. Imposing a symmetry condition on the jet by using a one-quarter domain model significantly changes the results, but employing a subgrid scale model has little effect.

Acknowledgements

I would like to express my sincere gratitude to my advisors Professor Brian G. Thomas and Professor S. Pratap Vanka for their advising and support throughout the course of my graduate study for Master's degree.

My research project was financially supported by the National Science Foundation and the Continuous Casting Consortium. The computations for this project were done on the machines at the Computational Fluid Dynamic Lab and the Metals Processing Simulation Lab at the University of Illinois at Urbana-Champaign. The commercial finite volume software package FLUENT (V6.1) used in the flux layer simulations is provided by Fluid Dynamics International. The support of these organizations is gratefully appreciated.

I would also like to thank several colleagues and friends for their helpings. Quan Yuan, Dr. Sarma Rani, Dr. Gang Luo and Dr. Chad M. Winkler are acknowledged for their helping on CFD issues. Ya Meng, Dr. Chunsheng Li and Dr. Lifeng Zhang were always ready to help on issues of the continuous casting process. These friends' guidance and company have made my graduate study an enjoyable experience.

Last but not the least I am grateful to my parents and my sister to whom I owe all that I have accomplished.

Table of Contents

List of	Figures	viii
List of	Tables	XV
List of	Symbols	xvi
Chapte	er 1. Introduction	1
1.1	Figure	4
Chapte	er 2. Numerical Study of Flow and Heat Transfer in a Molten	
	Flux Layer	5
2.1	Introduction	5
2.2	Previous Work	6
2.3	Governing Equations and Solution Method	8
2.4	Code Validation	9
	2.4.1 Drag flow with temperature dependent viscosity	9
	2.4.2 Buoyant convection in a thin layer	10
	2.4.3 Steady Rayleigh-Bernard convection	11
2.5	Details of the Simulations	12
	2.5.1 Flux viscosity	14
	2.5.2 Flux thermal properties	16
2.6	Results and Discussion	16
	2.6.1 Grid refinement study	16
	2.6.2 Effect of temperature dependent flux viscosity	16
	2.6.3 Effect of flux layer thickness	18
	2.6.4 Heat transfer rates	18
	2.6.5 Effect of bottom shear velocity	20
2.7	Summary	24
2.8	Figures and Tables	25
Chapter 3. Large Eddy Simulation of Heat Transfer by a Circular		
	Impinging Jet	48
3.1	Introduction	48
3.2	Literature Review	
3.3	Numerical Method	

	3.3.1	Governing equations	53
	3.3.2	Numerical method	56
	3.3.3	Initial and boundary conditions	57
	3.3.4	Computational details	58
3.4	Result	ts and Discussions	59
3.5	Summ	nary	62
3.6	Figure	es and Tables	62
Chapt	er 4. L	arge Eddy Simulation of Transient Flow and Heat Trans-	
		fer in the Mold Region	75
4.1	Introd	luction	75
4.2	Revie	w of Literature	77
4.3	Gover	rning Equations and Numerical Method	80
	4.3.1	Governing equations	81
	4.3.2	Initial and boundary conditions	81
	4.3.3	Computational details	82
4.4	Result	ts and Discussion	83
	4.4.1	Instantaneous velocity and temperature fields $\ldots \ldots \ldots$	84
	4.4.2	Mean velocity and temperature fields	88
	4.4.3	RMS statistics of velocity and temperature field $\ .$	90
	4.4.4	Comparison of velocity profile with dye injection experiment $% \mathcal{L}^{(n)}$.	91
	4.4.5	Comparison of temperature profile with plant measurements $% \left({{{\bf{n}}_{{\rm{s}}}}} \right)$.	92
	4.4.6	Heat transfer rate to the solidifying shell $\ldots \ldots \ldots \ldots$	93
	4.4.7	Effect of sub-grid scale model on the solution \ldots	95
	4.4.8	Superheat budget	97
4.5	Summ	nary	101
4.6	Figure	es and Tables	102
Chapt	er 5. (Conclusions	186
5.1	Flow	and Heat Transfer in Molten Flux Layer	186
5.2	Impin	ging Jet Heat Transfer	186
5.3	Flow a	and Heat Transfer in Liquid Pool of Continuous Casting of Steel	187
5.4	Futur	e Work	188
Refere	nces		189

List of Figures

Fig. 1.1	Schematic of the continuous casting process	4
Fig. 2.1	Schematic of mold flux layer	25
Fig. 2.2	Simulation domain and boundary conditions	25
Fig. 2.3	Velocity and temperature profile for drag flow with variable viscosity	26
Fig. 2.4	Velocity and temperature profile for buoyant convection in a thin	
	layer	26
Fig. 2.5	Velocity and temperature fields of Rayleigh-Benard convection	27
Fig. 2.6	Contours of vertical density gradient	27
Fig. 2.7	Contours of horizontal density gradient	27
Fig. 2.8	Change of Nu with respect to Ra	28
Fig. 2.9	Flux viscosity curves used for variable viscosity study	28
Fig. 2.10	Viscosity of two real fluxes (measured points) and three models ap-	
	proximations (lines) \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots	29
Fig. 2.11	Temperature and velocity profile from grid refinement study ($x =$	
	$0.695 m$, viscosity curve a and $U_b = 0.01 m \cdot s^{-1}$)	29
Fig. 2.12	Flow, temperature and viscosity field of liquid flux layer with vari-	
	able viscosity $(B=10000)$	30
Fig. 2.13	Flow, temperature and viscosity field of liquid flux layer with vari-	
	able viscosity $(B=23880)$	31
Fig. 2.14	Flow, temperature and viscosity field of liquid flux layer with vari-	
	able viscosity ($B=23880$, modified thickness, $Ra=5,000$)	32
Fig. 2.15	Heat flow increase as a function of convection strength $\ldots \ldots$	33
Fig. 2.16	Velocity field (top), temperature field (middle) and Viscosity field	
	(bottom) of liquid flux layer with variable viscosity (a) and bottom	
	shear velocity $0.1 m \cdot s^{-1}$	34
Fig. 2.17	Velocity field (top), temperature field (middle) and Viscosity field	
	(bottom) of liquid flux layer with variable viscosity (b) and bottom	
	shear velocity $0.1 m \cdot s^{-1}$	35
Fig. 2.18	Velocity field (top), temperature field (middle) and Viscosity field	
	(bottom) of liquid flux layer with variable viscosity (c) and bottom	
	shear velocity $0.1 m \cdot s^{-1}$	36
Fig. 2.19	Nu distribution on the top and bottom of flux layer (flux a) \ldots	37
Fig. 2.20	Velocity profile across the domain thickness (CL; flux a) \ldots .	37

Fig. 2.22Velocity profile across the domain thickness (CL; flux c)Fig. 2.23Temperature profile across the domain thickness (CL; flux a)	. 38 . 39 . 39
Fig. 2.23 Temperature profile across the domain thickness (CL; flux a) \ldots	. 39 . 39
	. 39
Fig. 2.24 Temperature profile across the domain thickness (CL; flux b)	
Fig. 2.25 Temperature profile across the domain thickness (CL; flux c) \ldots	. 40
Fig. 2.26 Viscosity profile across the domain thickness (CL; flux a)	. 40
Fig. 2.27 Viscosity profile across the domain thickness (CL; flux b)	. 41
Fig. 2.28 Viscosity profile across the domain thickness (CL; flux c) \ldots	. 41
Fig. 2.29 Effect of flux viscosity on the velocity profile $(U_b = 0.06 m \cdot s^{-1})$.	. 42
Fig. 2.30 Effect of flux viscosity on the velocity profile (interfacial shear stress	3
$5 N \cdot m^{-2}$)	. 42
Fig. 2.31 Relationship between interfacial shear stress and flux/steel velocity	43
Fig. 2.32 Nu number as a function of bottom shear velocity	. 44
Fig. 2.33 Nu number as a function of steel velocity	. 44
Fig. 3.1 Schematic of an impinging jet	. 62
Fig. 3.2 Simulation domain and boundary conditions	. 63
Fig. 3.3 Grid used in simulation for Reynolds number of 10^5	. 63
Fig. 3.4 Instantaneous velocity field $(Re=10^5)$. 64
Fig. 3.5 Instantaneous temperature field $(Re=10^5)$. 65
Fig. 3.6 Mean velocity field $(Re=10^5)$. 66
Fig. 3.7 Mean temperature field $(Re=10^5)$. 66
Fig. 3.8 Radial mean velocity profile at $r=1.0D$. 67
Fig. 3.9 Development of radial mean velocity profile $(Re=10^5)$. 67
Fig. 3.10 RMS axial velocity profiles $(Re=2\times 10^5)$. 68
Fig. 3.11 RMS radial velocity profiles $(Re=2\times10^5)$. 68
Fig. 3.12 RMS azimuthal velocity profiles $(Re=2\times10^5)$. 69
Fig. 3.13 RMS temperature profiles $(Re=2\times10^5)$. 69
Fig. 3.14 Effect of Reynolds number on RMS statistics	. 70
Fig. 3.15 Instantaneous Nu distribution $(Re=10^5)$. 71
Fig. 3.16 Radial distribution of mean and RMS Nu for $Re=5 \times 10^4$. 72
Fig. 3.17 Radial distribution of mean and RMS Nu for $Re=10^5$. 72
Fig. 3.18 Radial distribution of mean and RMS Nu for $Re=2 \times 10^5$. 73
Fig. 4.1 Equipment used in measuring temperature in the liquid pool	. 102
Fig. 4.2 Example of temperature measurement data	. 103
Fig. 4.3 Simulation domain and and boundary conditions	. 104

Fig. 4.4	Grids in the nozzle	105
Fig. 4.5	Grids in the mold	106
Fig. 4.6	Instantaneous velocity vector plot of center plane (with SGS model)	107
Fig. 4.7	Instantaneous T field of center plane (with SGS model) $\ldots \ldots$	108
Fig. 4.8	Instantaneous velocity and temperature field of center plane between	
	narrow faces (with SGS model)	109
Fig. 4.9	Instantaneous velocity and temperature field of cross section $17 mm$	
	from NF (with SGS model)	109
Fig. 4.10	Instantaneous velocity and temperature field of cross section 93 mm	
	to NF (with SGS model)	110
Fig. 4.11	Instantaneous velocity and temperature field of cross section $192 mm$	
	to NF (with SGS model)	110
Fig. 4.12	Instantaneous velocity and temperature field of cross section 291 mm	
	to NF (with SGS model)	111
Fig. 4.13	Instantaneous velocity and temperature field of cross section $38.5 mm$	
	from the top surface (with SGS model)	112
Fig. 4.14	Instantaneous velocity and temperature field of cross section $82 mm$	
	to the top surface (with SGS model)	112
Fig. 4.15	Instantaneous velocity and temperature field of cross section $161 mm$	
	to the top surface (with SGS model)	113
Fig. 4.16	Instantaneous velocity and temperature field of cross section $243 \ mm$	
	to the top surface (with SGS model)	113
Fig. 4.17	Instantaneous velocity and temperature field of cross section 339 mm	
	to the top surface (with SGS model)	114
Fig. 4.18	Instantaneous velocity and temperature field of cross section 445 mm	
	to the top surface (with SGS model)	114
Fig. 4.19	Instantaneous velocity and temperature field of cross section 741 mm	
	to the top surface (with SGS model)	115
Fig. 4.20	Instantaneous velocity and temperature field of cross section $944.5 mm$	
	to the top surface (with SGS model)	115
Fig. 4.21	Time averaged velocity field (with SGS model)	116
Fig. 4.22	Streamlines of mean velocity field (with SGS model)	117
Fig. 4.23	Mean velocity field of center plane between narrow faces (with SGS	
	model)	118
Fig. 4.24	Mean velocity field of cross section parallel to the narrow face (291 mm	
	from narrow face, with SGS model)	118

Fig. 4.25	Mean velocity field of cross section parallel to the narrow face $(192 mm)$	
	from narrow face, with SGS model)	119
Fig. 4.26	Mean velocity field of cross section parallel to the narrow face (93 mm	
	from narrow face, with SGS model)	119
Fig. 4.27	Mean velocity field of cross section parallel to the narrow face $(17 mm)$	
	from narrow face, with SGS model)	120
Fig. 4.28	Mean velocity field of cross section parallel to the narrow face $(0.35 mm)$	ı
	from narrow face, with SGS model)	120
Fig. 4.29	Mean velocity field of cross section parallel to the top surface (38.5 mm	
	from top surface, with SGS model)	121
Fig. 4.30	Mean velocity field of cross section parallel to the top surface (82 mm	
	from top surface, with SGS model)	121
Fig. 4.31	Mean velocity field of cross section parallel to the top surface (161 mm	
	from top surface, with SGS model)	122
Fig. 4.32	Mean velocity field of cross section parallel to the top surface (243 mm	
	from top surface, with SGS model)	122
Fig. 4.33	Mean velocity field of cross section parallel to the top surface (339 mm	
	from top surface, with SGS model)	123
Fig. 4.34	Mean velocity field of cross section parallel to the top surface (445 mm	
	from top surface, with SGS model) $\ldots \ldots \ldots \ldots \ldots \ldots$	123
Fig. 4.35	Mean velocity field of cross section parallel to the top surface (741 mm	
	from top surface, with SGS model)	124
Fig. 4.36	Mean velocity field of cross section parallel to the top surface (944.5 mr	n
	from top surface, with SGS model)	124
Fig. 4.37	3-D stream traces of the time-averaged flow field	125
Fig. 4.38	Mean temperature field of center plane (with SGS model) $\ . \ . \ .$	126
Fig. 4.39	Mean temperature fields in cross sections parallel to the narrow face	
	(with SGS model)	127
Fig. 4.40	Mean temperature fields in cross sections parallel to the top surface	
	(with SGS model)	128
Fig. 4.41	RMS u velocity in the center plane (with SGS model) $\ldots \ldots$	129
Fig. 4.42	RMS u velocity in the cross sections parallel to the narrow face (with	
	SGS model)	130
Fig. 4.43	RMS u velocity in the cross sections parallel to the top surface (with	
	SGS model)	131
Fig. 4.44	RMS v velocity in center plane (with SGS model) $\ldots \ldots \ldots$	132

Fig. 4.45	RMS v velocity in the cross sections parallel to the narrow face (with SGS model)	133
Fig. 4.46	RMS v velocity in the cross sections parallel to the top surface (with SGS model)	134
Fig. 4.47	RMS w velocity in center plane (with SGS model)	135
Fig. 4.48	RMS w velocity in the cross sections parallel to the narrow face (with SGS model)	136
Fig. 4.49	RMS w velocity in the cross sections parallel to the top surface (with SGS model)	137
Fig. 4.50	RMS temperature in the center plane (with SGS model)	138
Fig. 4.51	RMS temperature in the cross sections parallel to narrow face (with SGS model)	139
Fig. 4.52	RMS temperature in the cross sections parallel to the top surface (with SGS model)	140
Fig. 4.53	Mean flow field of simulation overlapped on the dye injection exper- iment (without SGS model)	141
Fig. 4.54	Mean flow field of simulation overlapped on the dye injection exper- iment (with SGS model)	142
Fig. 4.55	Velocity magnitude along jet axis compared with dye injection mea-	143
Fig 4 56	Top surface velocity compared with dve injection measurements	143
Fig. 4.57	Comparison of simulation mean temperature profile and plant mea- surement (measurement 1)	144
Fig. 4.58	Comparison of simulation mean temperature profile and plant mea- surement (measurement 2)	144
Fig. 4.59	Comparison of simulation mean temperature profile and plant mea- surement (measurement 3)	145
Fig. 4.60	Comparison of simulation mean temperature profile and plant mea- surement (measurement 4)	145
Fig. 4.61	Comparison of simulation mean temperature profile and plant mea- surement (measurement 5)	146
Fig. 4.62	Instantaneous heat fluxes through three solidifying faces (with SGS model)	1/7
Fig. 4.63	Time averaged heat fluxes through three solidifying faces (with SGS model)	148

Fig. 4.64	Heat fluxes fluctuations through three solidifying faces (with SGS	
	$model) \dots \dots \dots \dots \dots \dots \dots \dots \dots $	149
Fig. 4.65	Heat flux along the center line of narrow face	150
Fig. 4.66	Heat flux along the line in wide face $(20 mm \text{ from narrow face})$.	150
Fig. 4.67	Heat flux in the wide faces along the side jet	151
Fig. 4.68	Heat flux in the wide faces along the center jet $\ldots \ldots \ldots \ldots$	151
Fig. 4.69	Mean velocity field of the side nozzle port (without SGS model) $$.	152
Fig. 4.70	Mean velocity field of the side nozzle port (with SGS model) $\ . \ .$	153
Fig. 4.71	Instantaneous velocity vector plot in the center plane (without SGS	
	$model) \dots \dots \dots \dots \dots \dots \dots \dots \dots $	154
Fig. 4.72	Instantaneous temperature field in the center plane (without SGS	
	$model) \dots \dots \dots \dots \dots \dots \dots \dots \dots $	155
Fig. 4.73	Time averaged velocity field (without SGS model) $\ldots \ldots \ldots$	156
Fig. 4.74	Streamlines of mean velocity field (without SGS model) \ldots .	157
Fig. 4.75	Mean temperature field in the center plane (without SGS model) $% \mathcal{G}(\mathcal{G})$.	158
Fig. 4.76	Mean temperature fields in cross sections parallel to the narrow face	
	(without SGS model) $\ldots \ldots \ldots$	159
Fig. 4.77	Mean temperature fields in cross sections parallel to the top surface	
	(without SGS model) $\ldots \ldots \ldots$	160
Fig. 4.78	Mean velocity and temperature profile in the center plane at $x =$	
	0.1m	161
Fig. 4.79	Mean velocity and temperature profile in the center plane at $x =$	
	0.2m	162
Fig. 4.80	Mean velocity and temperature profile in the center plane at $x =$	
	0.3m	163
Fig. 4.81	Mean velocity and temperature profile in the center plane at $x =$	
	0.4m	164
Fig. 4.82	RMS u velocity in the center plane (without SGS model) \ldots	165
Fig. 4.83	RMS u velocity in the cross sections parallel to the narrow face	
	(without SGS model)	166
Fig. 4.84	RMS u velocity in the cross sections parallel to the top surface	
	(without SGS model)	167
Fig. 4.85	RMS v velocity in the center plane (without SGS model) \ldots	168
Fig. 4.86	RMS v velocity in the cross sections parallel to the narrow face	
	(without SGS model)	169

Fig. 4.87	RMS v velocity in the cross sections parallel to the top surface (with-	
	out SGS model)	170
Fig. 4.88	RMS w velocity in the center plane (without SGS model) \ldots .	171
Fig. 4.89	RMS w velocity in the cross sections parallel to the narrow face	
	(without SGS model) $\ldots \ldots \ldots$	172
Fig. 4.90	RMS w velocity in the cross sections parallel to the top surface	
	(without SGS model) $\ldots \ldots \ldots$	173
Fig. 4.91	RMS temperature in the center plane (without SGS model) \ldots .	174
Fig. 4.92	RMS temperature in the cross sections parallel to narrow face (with-	
	out SGS model)	175
Fig. 4.93	RMS temperature in the cross sections parallel to the top surface	
	(without SGS model) $\ldots \ldots \ldots$	176
Fig. 4.94	RMS velocity in the cross section parallel to the narrow face $(0.35mm$	
	from narrow face, without SGS model)	177
Fig. 4.95	RMS velocity in the cross section parallel to the narrow face $(0.35mm$	
	from narrow face, with SGS model)	178
Fig. 4.96	Instantaneous heat fluxes through three solidifying faces (without	
	SGS model)	179
Fig. 4.97	Time averaged heat fluxes through three solidifying faces (without	
	SGS model)	180
Fig. 4.98	Heat fluxes fluctuations through three solidifying faces (without SGS $$	
	model) \ldots	181
Fig. 4.99	System internal energy variation with time	182
Fig. 4.100	Superheat removal percentage distribution	183

List of Tables

Table 2.1	Parameters for simulation of drag flow with temperature dependent	
	viscosity	45
Table 2.2	Parameters for simulation of buoyant convection in a thin layer	45
Table 2.3	Parameters for validation cases against experiments	46
Table 2.4	Parameters for variable viscosity and thickness simulations \ldots .	46
Table 2.5	Parameters for shear velocity simulations	47
Table 3.1	Parameters for impinging jet simulation	74
Table 3.2	Grid details for simulations	74
Table 4.1	Plant experiments conditions	184
Table 4.2	Parameters and material properties for the simulation	185

List of Symbols

α	Thermal diffusivity $(m^2 \cdot s^{-1})$
eta	Volumetric expansion coefficient (K^{-1})
Δ	Grid length scale (m)
$\Delta_x, \Delta_y, \Delta_z$	Grid size in three coordinate directions (m)
μ_T	Turbulent viscosity $(kg \cdot m^{-1} \cdot s^{-1})$
μ	Molecular viscosity $(kg \cdot m^{-1} \cdot s^{-1})$
μ_0	Molecular viscosity at a reference temperature $(kg\cdot m^{-1}\cdot s^{-1})$
$ u_T$	Turbulent kinematic viscosity $(m^2 \cdot s^{-1})$
ν	Kinematic viscosity $(m^2 \cdot s^{-1})$
Φ	Scalar variable derived in the fractional step method
ρ	Density $(kg \cdot m^{-3})$
$ ho_0$	Density at a reference temperature $(kg \cdot m^{-3})$
C_P	Specific heat $(J \cdot kg^{-1} \cdot \mathbf{K}^{-1})$
g	Gravitational acceleration $(m \cdot s^{-2})$
k_C	Thermal conductivity $(W \cdot m^{-1} \cdot \mathbf{K}^{-1})$
K_G	sub-grid scale turbulent energy
k_R	Radiation thermal conductivity $(W \cdot m^{-1} \cdot \mathbf{K}^{-1})$
k_{eff}	Effective thermal conductivity $(W \cdot m^{-1} \cdot \mathbf{K}^{-1})$
p	Pressure $(Pa \cdot s)$
r	Radial distance from the impingement point (m)
t	Time (s)
u, v, w	Velocity components in x, y, z directions $(m \cdot s^{-1})$
u^*	Dimensionless velocity
U_B	Bulk jet velocity $(m \cdot s^{-1})$
U_b	Bottom shear velocity of liquid flux layer $(m \cdot s^{-1})$
V	Velocity scale $(m \cdot s^{-1})$
x,y,z	Coordinates in three directions (m)

x^*, y^*	Dimensionless coordinates
$x_i \ (i=1,3)$	Coordinates in three dimension (m)
ΔT	Temperature difference (K)
T	Temperature (K)
T_h, T_c	Constant temperature (K)
T_s	Solidification temperature of flux (K)
θ	Dimensionless temperature
$\bar{u}_i \ (i=1,3)$	Filtered velocity components for LES $(m \cdot s^{-1})$
$\bar{\tilde{u}}_i \ (i=1,3)$	Intermediate velocity components for LES $(m \cdot s^{-1})$
$u_i'\ (i=1,3)$	Subgrid-scale velocity fluctuation $(m \cdot s^{-1})$
$\bar{S}_{ij} \ (i,j=1,3)$	Filtered symmetric strain rate tensor
$Q_{ij} \ (i,j=1,3)$	Subgrid-scale momentum flux
$Q_{Ti} \ (i=1,3)$	Subgrid-scale heat flux
D	Jet diameter (m)
Н	Domain height in Chapter 2; Jet to plate distance in Chapter 3 (m)
L	Domain length (m)
Gr	Grashoff number
Nu	Nusslet number
Pr	Prandtl number
Ra	Rayleigh number
Ra_c	Critical Rayleigh number for onset of natural convection
Re	Reynolds number
C	Constant in Nu - Ra correlation
c_1	Constant in analytical solution for buoyant convection
C_{ε}	Constant in static k model
C_v	Constant in static k model
C_{kk}	Constant in static k model
E	Constant
R	Gas constant

AConstant in evaluating temperature dependent viscosityaConstant for evaluating temperature dependent viscosity (K⁻¹)BConstant in evaluating temperature dependent viscositynConstant in evaluating temperature dependent viscosity

Chapter 1. Introduction

The continuous casting process is the predominant manufacturing method in the steel industry; it is used to produce over 95% of the several hundred million tons of steel every year in the world. Although novel methods are being invented, continuous casting will still remain as the major process for mass production of steel in the foreseeable future. Continuous casting of steel is a very complex process, involving transport phenomena, solidification and multi-phase flow. Fig. 1.1 shows a representative schematic of the continuous casting process. Liquid steel is poured from a ladle into a tundish, and then flows through a ceramic submerged entry nozzle (SEN) and exits via bifurcated ports into the liquid steel pool in the mold. Steel in the mold solidifies against the water-cooled mold wall and forms a shell which holds the liquid steel pool. Out of the mold, rollers roll on the shell to drag it downward, the dragging speed is often referred to as "casting speed". The shell grows as it goes down and eventually solidifies through the complete thickness to form a solid slab.

Mold flux powder is added on top of the liquid steel pool. The flux powder will sinter and melt to form a liquid flux layer because it has a melting point $(1,100\sim1,500 \text{ K})$ lower than the temperature of the liquid steel (>1820 K). Fig. 1.1 shows layers of flux powder and liquid flux above the molten steel. Between the powder and liquid layer, there is also a layer of sintered flux. There are several important roles which the flux layer plays in the process. The flux layer provides thermal insulation for its low thermal conductivity. The flux layer provides chemical insulation because it prevents the steel from being oxidized by contacting with air. The flux layer also has the function of removing inclusions. It absorbs alumina particles, which are added in the nozzle to the atmosphere. Otherwise these inclusions will have a greater chance of being trapped in solidified steel. Finally, the liquid flux infiltrates into the gap between the solidifying steel shell and the mold, and acts as lubricant and promotes

uniform heat transfer in the mold strand gap. The mechanism of heat transfer in the liquid flux layer is very complex. The large temperature difference across the liquid flux layer coupled with temperature dependent density may cause buoyancy driven convection, which will increase the heat transfer rate through the liquid flux layer. The shear along the steel/flux interface induces forced convection and affects the heat transfer rate. Radiation should also be taken into consideration as the liquid flux layer is semitransparent.

The flow of liquid steel is highly turbulent, the Reynolds number (based on the nozzle diameter) is on the order of 100,000. Flow through the SEN is driven by gravity for there is a height difference between the liquid steel level in the tundish and the free surface in the mold. The flow rate of molten steel is usually controlled either by a "stopper rod" or by a "slide gate". The presence of these flow control devices will strongly affect the flow pattern in the nozzle as well as in the mold. Argon gas is injected through pores in the nozzle wall to prevent clogging. The presence of argon bubbles makes the flow multi-phase and the amount of argon gas has great influences over the steel flow pattern.

Through the bifurcated nozzle port, liquid steel forms a jet which flows across the liquid pool and impinges on the solidifying shell at the narrow face (NF). There is locally very high heat transfer rate due to the oblique impingement of the jet. The jet carries superheat, and has a shell thinning effect by eroding the shell where it impinges locally. Under extreme conditions, the thin shell will not be strong enough to hold the liquid steel and a costly "breakout" will happen. So a more spread jet and hence more evenly distributed heat transfer rate is preferred. After impingement, the jet bifurcates into an upper wall jet and a lower wall jet. The upward flowing wall jet travels towards the free surface, forms a recirculation region usually referred to as "lower roll". This flow pattern can change radically with different operating

conditions, like increasing argon injection rate, applying electromagnetic forces to brake and stir the liquid, different SEN and mold geometries or different casting speed. The flow pattern can fluctuate with time, leading to defects, so transient behavior is important. The flow pattern also affects the trajectory of inclusions and argon bubbles and causes quality problems such as slivers and pencil-pipes. Thus understanding the detailed transient fluid flow and heat transfer phenomena is of great importance to defect reduction in continuous casting.

The works of McDavid and Thomas^[1] and Sivaramakrishnan^[2] are the few studies on the heat transfer through the liquid flux layer. The effect of natural convection, bottom shear from the steel flow beneath coupled with strongly temperature dependent viscosity of liquid flux has not been thoroughly investigated so far. Many of the previous numerical studies on the flow and heat transfer in the mold region have been carried out using the Reynolds-averaged approach, which is only able to calculate mean flow and temperature field. However, many of the phenomena which lead to defect formation are transient and cannot be predicted by a steady state approach. Among the various numerical methods to study transient fluid flow and heat transfer, Large Eddy Simulation (LES) is a prominent method. There are some recent works^[2] which investigated transient fluid flow in the mold region using LES, but there is no work in applying LES to the heat transfer simulation in continuous casting so far.

The present work aims to study the fluid flow and heat transfer in different parts of the continuous casting process, in an attempt to better understand the process and defect formation. First the fluid flow and heat transfer in the liquid flux layer are simulated using a steady state approach, in the presence of natural convection, temperature dependent viscosity and shear from the steel flow beneath. Parametric studies have been conducted to simulate a wide range of flux properties and casting conditions. Next the circular air impinging jet heat transfer is simulated using LES as a validation of the numerical method. Then the transient fluid flow and heat transfer in the liquid steel pool mold region of a real continuous caster has been studied using the same numerical model used in the impinging jet study. The results of the simulation in the form of the mean flow, turbulent statistics and flow transients are analyzed and compared with plant measurement and water model experiment.

1.1 Figure



Fig. 1.1: Schematic of the continuous casting process

Chapter 2. Numerical Study of Flow and Heat Transfer in a Molten Flux Layer

2.1 Introduction

Slag floats on the surface of molten metals during many different processing and refining operations, including furnaces, ladles, tundishes and molds. In addition to assisting with chemical reactions, inclusion removal and protection from air absorption, this layer plays an important role in providing thermal insulation. One process where the liquid slag layer is particularly important is in the continuous casting of steel. A carefully designed mixture of oxides is added as a powder to the top of the molten steel at regular time intervals, where it sinters and melts to form a liquid flux layer that floats above the molten metal surface. The melting and reaction rates depend on the composition (especially carbon content), porosity, and thermal properties of the flux powder. The interface between the liquid flux and the sintered solid powder floating above is at the melting temperature of the flux and is generally quite rigid or viscous relative to the liquid below. The lower surface of the liquid flux layer is at the temperature of the molten steel flowing below it, and is subjected to shear velocity and shape changes which depend on the turbulent flow conditions in the molten steel. The liquid flux is drawn into the gap between the solidifying steel shell and the mold to provide lubrication and thermal uniformity. Fig. 2.1 shows a vertical cross section of the continuous casting process.

Superficially, the heat loss through the flux layers increases with increasing conductivity and decreasing layer thickness. However, heat transfer through the liquid flux layer actually occurs by conduction, natural convection, forced convection and even radiation for many fluxes which are semitransparent. Quantifying this heat transfer is important yet has received little attention in previous literature. It depends on many complex interacting factors, including the powder and flux properties (viscosity, conductivity, density, specific heat, and latent heat), vessel geometry, the shape and thickness of the powder and liquid flux layers, interfacial level fluctuations, and the bottom shear velocity imparted by the flowing metal below. The present work investigates this coupled fluid flow and heat transfer in the thin liquid flux layer using computational models, focusing on the effects of temperature-dependent viscosity, layer thickness, and bottom shear velocity. The results of this work will be useful for the prediction of heat transfer through flux and slag layers.

2.2 Previous Work

McDavid and Thomas^[1] performed one of the few computational studies of flow in flux layers. They simulated three-dimensional (3-D) steady, coupled fluid flow and heat transfer in the powder, liquid, and re-solidified flux layers using the finite element package, FIDAP. The steel-liquid flux interface velocity was found by iterating with a 3-D k- ε turbulent model of fluid flow in the nozzle and mold region of the continuous caster, until equal shear stress along the interface was achieved. The steel-flux interface shape and the rate of flux infiltration into the mold-strand gap were fixed to values measured in an operating steel caster. Temperature dependent properties were used. The converged solution matched the measured liquid flux layer thickness profile. The flow solution showed a single large recirculation region whose depth increased with increasing liquid flux conductivity and decreasing flux viscosity, owing to increased heat transfer across the layer. Like flow in the steel pool beneath it, flow within the liquid flux layer was predominantly in the plane normal to the narrow face as flux consumption had little effect on the flow pattern over most of the domain. The simulation showed that viscosity plays an important role in the thermal and flow behavior of the flux. This work performed only two simulations, and ignored natural convection effects.

Natural convection in the liquid flux layer arises because the density of the liquid flux is temperature dependent. The lower surface of the liquid flux layer is just above steel melting temperature (~ 1550 °C) while its top surface is at the melting temperature of the flux (800 ~ 1200 °C).^[1] This large temperature gradient causes a density gradient. The unstable system breaks down as buoyancy forces set up alternate rising and falling plumes, which transport hot low-density fluid upward and cold high density fluid downward. These merge together to form Rayleigh-Benard convection cells. These natural convection cells will increase the mixing and heat transfer rate significantly beyond pure conduction. The resulting fluid flow and heat transfer in these large aspect ratio fluid layers has been studied extensively.^[3-7] In the steel caster where the Rayleigh number is small, it is appropriate to assume that the fluid flow is two-dimensional with the cell axis along the width.

Booker^[8] measured heat transfer and studied convection cell structure in a high Prandtl number fluid between horizontal flat plates. The viscosity varied up to 300fold between the top and bottom boundary temperatures. The Nusselt numbers were 12% lower than predictions of standard correlations^[4] with the viscosity evaluated at the mean of the boundary temperatures. Mohamad and Viskanta^[9] computed 2-D laminar flow in a shallow cavity (0.1 aspect ratio) driven by surface-shear and buoyancy for a low Prandtl number fluid. The cavity was heated from below and cooled at the top, where the shear velocity was applied. The equations were solved using a finite-volume method with SIMPLE.^[10] The results showed that the shear velocity has an insignificant effect on the heat transfer when natural convection dominates, $(Gr/Re^2 \gg 1)$. Increasing shear velocity lowers heat transfer when $Gr/Re^2 \sim O(1)$ but increases heat transfer at higher velocities when forced convection dominates. Other simulations also showed that shear modified the Rayleigh-Benard convective cells generated due to heating from below.^[2] Recently, Sivaramakrishnan^[2] studied the transition between natural and forced convection flow in the liquid flux layer, using the finite-element program FIDAP. Above a critical bottom shear velocity, the natural convection cells are annihilated, and the flow pattern transforms into a single large recirculation region with a lower heat transfer rate. Parametric studies on flux viscosity and bottom shear velocity were performed, but this study did not include temperature dependent properties.

The present study computes fluid flow and coupled heat transfer in a liquid flux layer, accounting for the combined effects of natural convection, bottom shear velocity and strongly temperature dependent viscosity. Computations are performed for several different commercial fluxes and bottom shear velocities in a rectangular domain, shown in Fig. 2.2.

2.3 Governing Equations and Solution Method

To compute the fluid flow and heat transfer in this problem, the steady Navier-Stokes equations including buoyant body forces are solved for mass continuity, momentum in x and y direction and the heat balance:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2.1}$$

$$\frac{\partial(uu)}{\partial x} + \frac{\partial(vu)}{\partial y} = -\frac{1}{\rho_0}\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(\nu\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\nu\frac{\partial u}{\partial y}\right) + \frac{\partial\nu}{\partial x}\frac{\partial u}{\partial x} + \frac{\partial\nu}{\partial y}\frac{\partial v}{\partial x}$$
(2.2)

$$\frac{\partial(uv)}{\partial x} + \frac{\partial(vv)}{\partial y} = -\frac{1}{\rho_0}\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left(\nu\frac{\partial v}{\partial x}\right) + \frac{\partial}{\partial y}\left(\nu\frac{\partial v}{\partial y}\right) + \frac{\partial\nu}{\partial x}\frac{\partial u}{\partial y} + \frac{\partial\nu}{\partial y}\frac{\partial v}{\partial y} + \beta\Delta Tg$$
(2.3)

$$\frac{\partial(uT)}{\partial x} + \frac{\partial(vT)}{\partial y} = \frac{\partial}{\partial x} \left(\alpha \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(\alpha \frac{\partial T}{\partial y} \right)$$
(2.4)

where $\nu = \mu(T)/\rho_0$ is the kinematic viscosity, $\alpha = k_{eff}/\rho_0 C_p$ is the thermal diffusivity and $\Delta T = T - T_0$ is the temperature difference. It is assumed that flow in the flux layer is predominantly two-dimensional and laminar. The buoyancy effect is modelled by an extra term in the y-momentum equation (gravity direction) according to the Boussinesq approximation.^[11]

The above equations are solved with the commercial fluid flow package FLUENT, version 6.1.^[12] The discretization scheme used is second order upwind for momentum and energy equations, and the SIMPLE scheme for pressure-velocity coupling. The steady state equations are solved using the segregated solver. The convergence criterion for all the simulations was 10^{-6} , which means that the scaled residual of the final solution is reduced to 10^{-6} of the initial residual (defined at the fifth iteration).

2.4 Code Validation

The code was first validated against analytical solutions in several different problems, including flow between two parallel plates with temperature-dependent viscosity and buoyant convection in a thin layer. Excellent agreement was achieved in all cases. The code was further validated against experimental data involving steady Rayleigh-Benard convection in large-aspect ratio cavities.

2.4.1 Drag flow with temperature dependent viscosity

The code is first validated against analytical solution of a drag flow between parallel plates with temperature dependent viscosity. The computational domain is the same as that in Fig. 2.2 but with different boundary conditions. The plates are held at different temperatures. The upper wall is moving with velocity U and the bottom wall is held stationary. The lateral boundaries are periodic. The viscosity of the fluid is a function of temperature which has the form of $\mu = \mu_0 e^{-a(T-T_0)}$, where μ_0 and a are material constant and T_0 is a reference temperature at which the viscosity equals μ_0 . The density of fluid is constant and there is no pressure gradient acting on the fluid.

The velocity and temperature profile can be solved analytically^[13] by assuming the flow is two dimensional, fully developed and Newtonian. The analytical solution to this problem can be given by the flowing equations.^[13]

$$T - T_0 = (T_1 - T_0)\frac{y}{H}$$
(2.5)

$$\frac{u}{V} = \frac{e^{a(T_1 - T_0)y/H} - 1}{e^{a(T_1 - T_0)} - 1}$$
(2.6)

The mesh used in the FLUENT simulation has 320 cells in the length direction and 32 cells in the thickness direction. Parameters and constants used in the simulation are given in Table 2.1. In the simulation, the domain was tilted by 45° to preserve both viscosity gradients in x and y direction, which provides a more strict test case.

Figure 2.3 shows the comparison of temperature and velocity profile of simulation results versus analytical solution. The numerical results match the analytical solution very well. The RMS error of temperature is 0.047 K and the RMS error of velocities is $4.7 \times 10^{-4} \, m \cdot s^{-1}$.

2.4.2 Buoyant convection in a thin layer

The buoyant convection flow in a thin layer, which has an analytical solution, was used to validate the code further. Again, the simulation domain is similar with that in Fig. 2.2 but with different boundary conditions. The top and bottom plates are stationary and insulated. The right wall is maintained at a higher constant temperature T_h while the left wall is maintained at a lower temperature T_c . Gravity acts in the layer thickness direction. The material properties of the fluid are all constant. Boussinesq approximation is assumed to account for the buoyancy force. The fluid is Newtonian and the velocity and temperature distributions are steady and two-dimensional.

Away from the ends, the flow is fully developed and has an analytical solution.^[13] The analytical solution can be described by the following equations:^[13]

$$u^* = \frac{c_1}{6} (y^{*3} - y^*) \tag{2.7}$$

$$\theta = c_1 x^* + \frac{c_1^2 Ra}{360} \frac{H^2}{L^2} (3y^{*5} - 10y^{*3} + 15y^* + 8)$$
(2.8)

$$c_1 = \left(-1 + \sqrt{1 + \frac{8Ra}{45}\frac{H^2}{L^2}}\right) / \left(\frac{4Ra}{45}\frac{H^2}{L^2}\right)$$
(2.9)

where $x^* = x/L$, $y^* = y/H$, $\theta = (T - T_c)/(T_h - T_c)$ and $u^* = u/V$ are the dimensionless variables. $V = \rho_0 g \beta \Delta T H^3/(\mu L)$ is the velocity scale. $Ra = \rho_0 g \beta \Delta T H^3/(\mu \alpha_0)$ is the Rayleigh number.

The grid used in the FLUENT simulation has 320 cells in the length direction and 32 cells in the thickness direction. Table 2.2 gives the parameters and constants used in the simulation. Figure 2.4 shows the comparison between simulation results and analytical solutions. The numerical results match the analytical solution very well. The RMS error of temperature is 0.015 K. The RMS error of velocities is $1.2 \times 10^{-7} \, m \cdot s^{-1}$.

2.4.3 Steady Rayleigh-Bernard convection

Kirchartz and Oertel^[6] measured natural convection flow in a thin cavity with aspect ratio of 10:4:1 (length : width : height) that was heated from below. The top and bottom were copper plates kept at constant temperatures. The sidewalls were glass which has higher thermal conductivity than the fluid, silicon oil. The density variations produced by the temperature distribution were visualized using a differential interferogram. A two-dimensional simulation was performed to match the experimental conditions for Case 1, given in Table 2.3. The side walls were assumed to be perfectly conducting with a linear temperature gradient. The flow field and temperature contours obtained using a grid of 320 cells along the length and 32 cells in the height are shown in Fig. 2.5. The periodic high temperature gradients where the rising plumes impinge on the top surface cause increased local heat transfer rates. Figure 2.6 and Fig. 2.7 compare the fringes of the experimental interferograms with density gradients constructed from the temperature gradients using the relation $d\rho = -\beta dT$. Both experimental and computational results show ten natural-convection cells with aspect ratio about unity. The end cells are distorted due to interactions with the side walls. The simulated figures mirror the experimental figures, with the cells rotating in the opposite direction, as two symmetrical stable solutions are possible.

A series of similar simulations was then performed with adiabatic side walls by varying the liquid layer thickness. The Rayleigh number $(Ra = \rho g \beta \Delta T H^3 \mu^{-1} \alpha^{-1})$, characterizing the strength of the natural convection, and the fluid properties, included in the Prandtl number $(Pr = \nu/\alpha)$, are provided as Case 2 in Table 2.3. Figure 2.8 compares the simulated results with measurements by Rossby^[4] for the same conditions. The Nusselt number at the top surface $(Nu = H/\Delta T \cdot \partial T/\partial y)$ represents the heat transfer rate relative to pure conduction. The average Nu across the top surface increases linearly with the logarithm of Ra. There is slight deviation for Ra numbers above 10⁵, which may be due to the onset of oscillatory convection. The good agreement seen here validates the adequacy of this computational grid for the simulations in this study.

2.5 Details of the Simulations

The domain and boundary conditions used for computations of convection in a liquid flux layer are shown in Fig. 2.2. The flux layer is approximated as a two

dimensional rectangular domain with a length of 0.7 m and thickness of 0.01 m. This represents a slice through the vertical plane between the narrow faces of a 1.4 m wide continuous cast strand, extending from the left narrow face to the SEN center. The depth of the liquid flux layer develops from a balance between the melting of the powder, which depends on the rate of heat flow through the layer, and the consumption of liquid at the meniscus. This depth often varies from the narrow face to the submerged entry nozzle, according to the flow pattern of the molten steel beneath it. For this study, the flux layer thickness is assumed constant at a typical depth of 10 mm. The small effect of the flux infiltration into the mold-strand gap is neglected.

Because of the steep increase in viscosity, the top of the liquid layer is approximated as a flat surface at the flux melting temperature. The lower surface is set to the molten steel temperature. The value of $1550 \,^{\circ}$ C represents a typical superheat of about $5 \,^{\circ}$ C above the liquidus temperature of $1545 \,^{\circ}$ C for very-low carbon steel. The right side of the domain is a symmetry plane, so is an adiabatic, free-slip wall. The left side is in contact with the mold, which should be a wall at the flux solidification temperature. However, to avoid singularity at the left-bottom corner, and to represent the effects of flux leaking into the gap between the steel shell and the mold, the bottom half of the left wall is given a linear temperature profile. Table 2.4 gives the standard conditions and properties used in the simulations.

The shear velocity along the bottom steel flux interface is varied parametrically to investigate the effect on convection in the liquid flux layer. The steel velocity increases from about 0.05 to $0.4 \, m \cdot s^{-1}$ as casting speed increases.^[14] To match the interfacial shear stress, the liquid flux velocity is much smaller, owing to its higher viscosity. Specifically, the corresponding liquid flux velocity is about 1 to $65 \, mm \cdot s^{-1}$ for a typical flux with 0.2 *Pas* interface viscosity,^[15] based on balancing the interfacial shear stress according to previous work.^[1,2] The bottom velocity also varies from zero at the edges to a maximum midway between the narrow face and SEN.^[1] In this work, constant bottom shear velocities from 0 to $200 \, mm \cdot s^{-1}$ are assumed.

2.5.1 Flux viscosity

The viscosity of the liquid flux layer in a continuous caster varies greatly with its composition and temperature. Commercial fluxes typically contain mainly Al_2O_3 (0~ 13%), CaO (22 ~ 45%), and SiO_2 (17 ~ 56%),^[16] with small amounts of fluorides (NaF, CaF_2), alkalis (Na_2O , K_2O) and other basic oxides (MgO, BaO). Increasing the SiO_2 content enhances cross linking of the silicate chains, and thereby increases viscosity.^[17] Increasing Al_2O_3 content also increases the viscosity.^[17] As Al_2O_3 is continuously absorbed into flux layer from aluminum-deoxidized steel, its content increases up to 30 %. The viscosity of liquid flux decreases with temperature according to an Arrhenius equation.^[17]

$$\mu = A \exp(\frac{E}{RT}) \tag{2.10}$$

As the powder sinters, its viscosity increases greatly, exceeding $10^4 Pa \cdot s.^{[1]}$ The interface between the powder and the liquid often becomes crusty and enriched in carbon.^[18] Beneath this rigid interface, the viscosity decreases according to the increase in local temperature. Upon re-solidification against the mold at the meniscus (left domain wall), the flux viscosity increases according to the cooling rate and its crystallinity, but this effect is beyond the scope of the present work.

There are many empirical equations for flux viscosity. Riboud and Larrecq^[19] give one such an equation, based on temperature and composition.

$$\mu = ATe^{B/T} \tag{2.11}$$

$$\ln A = -20.81 - 35.75x_{Al_2O_3} + 1.73x_{CaO} + 5.82x_{CaF_2} + 7.02x_{Na_2O}$$
(2.12)

$$B = 31140 + 68833x_{Al_2O_3} - 23896x_{CaO} - 46351x_{CaF_2} - 39519x_{Na_2O}$$
(2.13)

where μ is viscosity in $Pa \cdot s$, T is temperature in Kelvin and x is the mole fraction of constituent compound. For a typical flux (45% SiO_2 , 10% Al_2O_3 , 10% CaO, 10% CaF_2 , 15% Na_2O), A is 5.6 to 10 $Pa \cdot s \cdot K^{-1}$ and B is ~24,000 K. To characterize a range of fluxes for a parametric study, Eq. 2.11 was transformed to:

$$\mu = \mu_0 \frac{T}{T_0} e^{B(1/T - 1/T_0)} \tag{2.14}$$

where T_0 is a reference temperature (1773 K), μ_0 is a reference viscosity (0.05 $Pa \cdot s$), and B is a parameter representing the degree of temperature dependency of the flux viscosity. Figure 2.9 shows various viscosity curves with this equation for different values of B, that represent the artificial fluxes simulated in this work.

This study also investigates two real industrial fluxes, commonly used in steel plants. Curves of the following form were fitted to measurements of flux viscosity taken from Larson^[15] and Lanyi and Rosa.^[20]

$$\mu = \mu_0 \left(\frac{T_0 - T_s}{T - T_s}\right)^n \tag{2.15}$$

where μ_0 is the viscosity at the reference temperature, T_0 of 1300 °C, and T_s is the fitting parameter.

Figure 2.10 shows the two viscosity curves. Viscosity curve (a)^[15] shows the typical behavior of a glassy flux, whose viscosity decreases smoothly with increasing temperature. Curve (b)^[20] depicts a typical crystalline flux, and was chosen to investigate behavior where the viscosity drops suddenly from the solid state upon melting. Except for missing the sharp peak near the melting point, the data for this crystalline flux is also reasonably approximated using Eq. 2.15 with B=23,880 K, which is shown as curve (c) in Fig. 2.10.

2.5.2 Flux thermal properties

Thermal conductivity of the liquid flux layer varies over a wide range. Since the liquid flux is semitransparent, radiation upward from the lower surface (liquid steel) causes additional heat transfer. This was approximated by defining a "radiation thermal conductivity" (k_R) .^[21,22] The conductivity (k_C) of liquid flux with only conduction is about $0.2 \sim 0.6 W \cdot m^{-1} \cdot K^{-1}$.^[21,23] Radiation increases this significantly, depending on the steel surface temperature and flux melting temperatures (which are known), and the flux absorption properties, which are estimated. A typical value of $3W \cdot m^{-1} \cdot K^{-1}$ is used for effective conductivity (k_{eff}) , as assumed elsewhere.^[1] The specific heat of the flux was assumed to be constant at $2000 J \cdot kg^{-1} \cdot K^{-1}$, as assumed elsewhere.^[1] Buoyancy is represented by the Boussinesq approximation, assuming a constant volumetric expansion coefficient β of $2.4 \times 10^{-5} \text{ K}^{-1}$.^[1] For typical temperature differences across flux layers, $\beta \Delta T \ll 1$ so this approximation is valid. Thus, a constant density of $2500 kg \cdot m^{-3}$ was adopted.

2.6 Results and Discussion

2.6.1 Grid refinement study

To investigate grid independence, simulations were performed for two grids, 640×32 and 1280×64 , assuming bottom shear velocity of $0.01 \ m \cdot s^{-1}$ and viscosity curve (a) in Fig. 2.10. Figure 2.11 compares the velocity and temperature profiles near the right wall, where the largest gradients are found. The results differ by less than 1%, so the 640×32 mesh is used for the remaining calculations.

2.6.2 Effect of temperature dependent flux viscosity

The effect of temperature dependent flux viscosity on natural convection was first studied with 13 simulations, based on Eq. 2.11 with B values ranging from 5,000 K

to 23,880 K. The assumed viscosity curves (Fig. 2.9) represent variations on a real flux ($B\sim24,000$ K) but with decreasing temperature dependency, and corresponding decreasing viscosity at the upper interface (1000 °C). A case with an extremely low constant viscosity of $0.05 Pa \cdot s$ was also performed. These simulations were done with zero bottom shear velocity in order to first study just the effect of viscosity variations. Other parameters, including domain thickness were constant, as given in Table 2.4.

The flow, temperature and viscosity fields of the flux layer with intermediate temperature variation of viscosity (B=10,000 K) is shown in Fig. 2.12. The results are generally similar to the classic Rayleigh-Benard convection pattern given in Fig. 2.5 for a constant viscosity. Slight differences arise at the domain ends, owing to the different boundary conditions at the side walls. However, these end effects are insignificant due to the large aspect ratio of the domain. More importantly, with constant viscosity, the upward and downward plumes have the same strength, so the velocity and temperature fields have the same shape when inverted. The viscosity variation weakens the convection, and the rising plumes are almost twice as wide as the falling plumes. It lowers the maximum velocity to $\sim 1 \, mm \cdot s^{-1}$, compared with $2 \, mms^{-1}$ with the constant viscosity case of 0.05 Pas. This damping of the convection reduces the temperature gradients and heat transfer. If the viscosity - temperature relationship was linear, then the viscosity variation with temperature fields would appear the same. However, the nonlinear viscosity variation with temperature also causes the appearance of the viscosity field to differ from the temperature field.

As the viscosity variation is increased (increasing B), the higher viscosity progressively weakens natural convection until the convection cells completely disappear. Figure 2.13 shows results for the strong viscosity variation of a typical real flux (B=23,880 K), where natural convection is suppressed. The velocity field is almost stagnant. The maximum velocity is only $\sim 0.1 \text{ mm} \cdot \text{s}^{-1}$, caused only by the boundary condition at the left wall. The nearly linear temperature field corresponds to pure conduction with the minor end effects.

2.6.3 Effect of flux layer thickness

Increasing thickness of the liquid flux layer promotes natural convection. The effect of increasing flux layer thickness was investigated for thicknesses between ~ $10 \, mm$ and ~ $20 \, mm$ for a real flux (curve c in Table 2.5 and Fig. 2.10) and other conditions given in Table 2.4. Figure 2.14 shows results for the same conditions as Table 2.4 except that the layer thickness is increased to $15.92 \, mm$. With the increased thickness, the natural convection is no longer completely suppressed. The maximum velocity increases to $1 \, mm \cdot s^{-1}$. The shape of the convection cells changes, as flow is restricted mainly to the lower part of the domain where the fluid is less viscous. Flow in the top portion is nearly stagnant. Each convection cell is smaller, and has a width to height aspect ratio of only 0.63, if the inactive region at the top of the cell is included.

2.6.4 Heat transfer rates

The rate of heat transfer through the flux layers is presented in Fig. 2.15 in terms of the average Nusselt number across the top surface as a function of the Rayleigh number. Results for the two sets of simulations (varying B and varying thickness) are compared with theoretical and experimental values for constant viscosity. The classic results for constant viscosity increase Ra by increasing domain thickness and / or temperature difference across the layer. For the set of simulations with varying temperature dependency, increasing Ra is obtained by decreasing the value of B, which decreases the average viscosity (while other parameters are constant, as given in Table 2.4). For the second set of simulations, the Ra is increased by increasing the domain thickness (while viscosity is held the same at B=23,880 K). As expected,
natural convection increases by a factor of 2 or 3 with increasing Ra. For a fluid with variable viscosity, a characteristic viscosity is needed to define the Ra number. Following Booker^[8] the Rayleigh number was evaluated here using the viscosity at the mean temperature of top and bottom boundaries (1275 °C).

Based on linear stability theory, the minimum Rayleigh number to start natural convection, Ra_c , is 1707 for a constant viscosity fluid.^[3] For the fluids with temperature dependent viscosity considered in this work, the critical Ra number was determined by fitting the simulation results to the following Nu-Ra correlation developed for constant-viscosity large-Pr fluids:^[3]

$$\frac{Ra(Nu-1)}{Ra-Ra_c} = C \tag{2.16}$$

This equation is only valid for Ra larger than Ra_c and less than about 3 Ra_c . The constant C increased from 1.43 for constant viscosity^[3] to 1.827 for the variable viscosity case. The critical Ra number, at the Nu=1 intercept (pure conduction), increased to 2,285. The higher critical Rayleigh number is consistent with the higher viscosity region suppressing natural convection cells. Basing the viscosity on the average temperature and using Eq. 2.16 with C=1.43 underestimates this critical Ra, which agrees with similar findings based on experiments by Booker.^[8] As Raincreases, the temperature dependency of the viscosity decreases, becoming closer to a constant viscosity, so the curve approaches the constant viscosity curve.

It is further seen that the critical Ra for changing flux layer thickness is 2,403 and the curve is translated to the right, relative to constant viscosity results. This critical Ra exceeds that of the other sets as expected, because it was based on the viscosity function with the largest value (B=23,880 K). The constant in Eq. 2.16 increases to 1.446. The curve based on temperature-dependent viscosity falls between the curves based on constant viscosity and variable layer thickness. These results demonstrate that Ra number alone does not provide sufficient information to characterize the natural convection strength for real fluids where viscosity varies with temperature.

2.6.5 Effect of bottom shear velocity

In a real steel caster, the liquid flux layer floating on top of the steel free surface in the mold is always subject to a shear velocity on its bottom surface. This shear velocity greatly affects flow and heat transfer in the liquid flux. Increasing this velocity causes the steel flux interface to become wavy, with flow and thickness variations known as "level fluctuations". If it becomes too large, liquid flux may be entrained into molten steel to form inclusion defects in the final product. The shear velocity also greatly affects heat conduction across the flux layer. In this work, we have investigated the effect of controlled shear velocity on the convection in the liquid flux layer. The bottom surface is assumed to remain flat with a constant domain thickness. Simulations are performed for the three viscosity curves for real fluxes given in Table 2.5 and Fig. 2.10.

The Rayleigh numbers calculated for viscosity curves (a), (b), and (c) (based on the viscosity at the average temperature) are 353, 1375 and 1239 respectively. The results in Fig. 2.15 show that the Ra numbers for all three cases are below the smallest critical Ra number, so no natural convection cells are expected. Applying shear velocity to the bottom surface further suppresses the formation of natural convection cells. The simulation results confirm this for all cases.

Figure 2.16 to Fig. 2.18 shows typical flow, temperature and viscosity fields for the liquid flux layer subjected to a bottom shear velocity of $0.1 \, m \cdot s^{-1}$. The results for all three viscosity curves are very similar. This shows that accurate modelling of the sharp increase in viscosity near the solidification temperature is not important, which is logical for very high viscosities. The flow fields share the common feature of one large recirculation region. The temperature fields feature larger temperature gradients at the right end of the domain caused by upward turning of the flow and almost pure conduction in the center region. This is because the flow is essentially stratified and laminar, so there is no vertical mixing or convective heat transfer.

In simulations shown in Fig. 2.12 and Fig. 2.14 where there is natural convection, the heat transfer rate varies in an oscillatory manner across the length of the top surface. The peaks and valleys correspond to upward and downward moving plumes. When the bottom shear velocity generates a single recirculation loop, the heat transfer across the domain is greatly skewed, with a large (negative) maximum towards the right wall on the top surface. The bottom surface has an even larger maximum at the lower left corner. These skewed distributions, shown in Fig. 2.19, should be considered, in addition to the average heat transfer.

The effect of bottom shear velocity on the horizontal velocity profile at the center of the domain is shown in Fig. 2.20 for viscosity curve (a). The interior velocities logically increase with bottom shear velocity. The flow direction changes at a height of 16-20% of the layer thickness. The height of this eddy center increases slightly with bottom shear velocity. Above 0.0086 m, the velocity diminishes to zero due to the high viscosity in this region. This makes the profile deviate from the parabolic profile of Couette flow. The vertical velocities are negligible, owing to the large aspect ratio of the domain. The horizontal velocity profiles for flux (b) and flux (c) are shown in Fig. 2.21 and Fig. 2.22, respectively. The velocity profiles share similar features of the profile for flux (a). However, the stagnation region at the top of the domain is the thickest for flux (a) and the height of eddy center is the largest for flux (b). Flux (b) has the smallest average viscosity and its viscosity only increases sharply in a very narrow temperature range, this viscosity profile makes it has the nearly no stagnation region and the highest eddy center.

The corresponding temperature profiles are shown in Fig. 2.23 to Fig. 2.25. With small bottom shear velocity, the flow at the domain ends does not affect flow near

the center, so the temperature profile is linear as in pure conduction. With increasing bottom shear velocity, the end effects extend towards the center and the temperature profile departs from linearity. Because their different viscosity profiles, the bottom shear velocities needed for the solution to deviate from conduction solution are different for the three fluxes. And also the extent the temperature profiles deviate from linearity is different for the three fluxes. Temperature profile for flux (b) deviates the most for it has the smallest average viscosity. Figure 2.26 to Fig. 2.28 show the viscosity profiles for the three fluxes, which vary nonlinearly according to the temperature.

The effect of different flux viscosity curves on the velocity profile is shown in Fig. 2.29 and Fig. 2.30. The relationship between shear stress and bottom velocity given in Fig. 2.31 is computed from these results. In addition to the flux viscosity profile, this relationship depends on thickness of the flux layer, $(10 \, mm \, here)$ and the interface temperature. The interface temperature is that of the molten steel, which is always around 1550 °C. This figure also shows the corresponding velocities in the steel near the top of the molten pool, just outside the boundary layer at the interface. These were computed using the logarithmic relationship for turbulent boundary layers.^[1,2]

For conditions of constant shear stress across the bottom surface, Fig. 2.30 shows that the velocities increase greatly from flux (a) to (b) to (c). This is due to the decreasing bottom surface viscosity, from flux curves (a) to (b) to (c). The average viscosity is actually lowest for flux (b), owing to its higher average temperature in the domain. The results are compared in Fig. 2.29 for constant bottom shear velocity. The height of the eddy center is highest for flux (b) (0.0024 m), owing to its viscosity profile (Fig. 2.10) which is relatively constant over most of the domain, except for the sharp increase at the very top. For the same reason, the stagnant region near the top is smaller with viscosity curve (b). The temperature profiles appear similar for all three fluxes and deviate only slightly from linearity for practical values of bottom shear velocity.

In the absence of natural convection cells, the flow pattern with a steady bottom shear velocity in this rectangular domain is stratified and there is no convective heat transfer, except near the ends. The rate of heat transfer, expressed in terms of Nunumber, is shown in Fig. 2.32 as a function of bottom shear velocity. It can be seen that Nu increases almost linearly with shear velocity. This is due to extension of the end effects with increasing circulation velocity in the domain. The rate of increase depends on the viscosity. The Nu number increases fastest for case (b), which has smallest average viscosity, and slowest for case (a), which has largest. This is because the end effect extends farther for the lower viscosity. The corresponding results with velocity in the molten steel are shown in Fig. 2.33. The trends are similar, except the curves increase logarithmically, owing to the nonlinear relationship between interface shear stress and velocity in the turbulent steel (Fig. 2.31).

The increase in heat transfer rate with interface shear velocity agrees with previous findings.^[1,2] It is expected from operating experience that increasing steel velocity increases heat transfer in the liquid flux pool. This in turn increases the melting rate at the flux / powder interface and increases the liquid flux layer thickness.

The maximum Nu is only about 2.3 for bottom shear velocity of $0.2 \, m \cdot s^{-1}$. Most liquid flux layers are subject to shear velocity less than $0.05 \, m \cdot s^{-1}$, where the Nu is less than 1.3. These small values show that forced convection from the bottom shear velocity produces only modest increases in heat transfer above the value for pure conduction. It is much smaller than the increase resulting from natural convection. This work suggests that plant observations of larger increases are likely due to phenomena not considered here. These include fluctuations and break-up of the bottom interface shape, caused by bubble motion, turbulent flow of the molten metal beneath the layer, and even slag emulsification. In addition, higher recirculation velocities in the domain may enhance mixing within the sintering flux above the top interface. This would increase the local kinetics of melting, resulting in higher effective conductivity, and a thicker liquid layer. Such phenomena should be investigated in future work.

2.7 Summary

Computational models are used to simulate 2-D fluid flow and heat transfer in the liquid flux layer above a molten metal surface, such as encountered in the continuous casting of steel. The model includes the effects of natural convection, temperature-dependent viscosity, and shear velocity across the bottom surface. It is found that the Ra number for realistic liquid slag layers varies near the critical Ra number for the onset of natural convection. For fluxes with temperature-dependent viscosity, the variation of Nu with Ra is analogous to correlations for fluids with constant viscosity evaluated at the mean temperature, but the critical Ra number is larger. The increase in Nu number with layer thickness is also quantified for realistic fluxes.

For thin layers of realistic fluxes, natural convection is suppressed, so Nu increases linearly with increase of bottom shear velocity. The increase is greater with decreasing average viscosity. The increase of heat transfer above pure conduction is only due to end effects, and hence depends on the dimensions of the layer. For the flat interface shape investigated here, this increase is only one to three fold. Larger increases observed in practice could be due to phenomena not included in these computations.

2.8 Figures and Tables



Fig. 2.1: Schematic of mold flux layer



Fig. 2.2: Simulation domain and boundary conditions



Fig. 2.3: Velocity and temperature profile for drag flow with variable viscosity



Fig. 2.4: Velocity and temperature profile for buoyant convection in a thin layer



Fig. 2.5: Velocity and temperature fields of Rayleigh-Benard convection



Fig. 2.6: Contours of vertical density gradient, above from numerical simulation, bottom from experiment $^{[6]}$



Fig. 2.7: Contours of horizontal density gradient, above from numerical simulation, bottom from $experiment^{[6]}$



Fig. 2.8: Change of Nu with respect to Ra



Fig. 2.9: Flux viscosity curves used for variable viscosity study



Fig. 2.10: Viscosity of two real fluxes (measured points) and three models approximations (lines)



Fig. 2.11: Temperature and velocity profile from grid refinement study $(x = 0.695 m, \text{ viscosity curve a and } U_b = 0.01 m \cdot s^{-1})$



Fig. 2.12: Flow, temperature and viscosity field of liquid flux layer with variable viscosity (B=10000)



Fig. 2.13: Flow, temperature and viscosity field of liquid flux layer with variable viscosity (B=23880)



Fig. 2.14: Flow, temperature and viscosity field of liquid flux layer with variable viscosity (B=23880, modified thickness, Ra=5,000)



Fig. 2.15: Heat flow increase as a function of convection strength



Fig. 2.16: Velocity field (top), temperature field (middle) and Viscosity field (bottom) of liquid flux layer with variable viscosity (a) and bottom shear velocity $0.1 \, m \cdot s^{-1}$



Fig. 2.17: Velocity field (top), temperature field (middle) and Viscosity field (bottom) of liquid flux layer with variable viscosity (b) and bottom shear velocity $0.1 \, m \cdot s^{-1}$



Fig. 2.18: Velocity field (top), temperature field (middle) and Viscosity field (bottom) of liquid flux layer with variable viscosity (c) and bottom shear velocity $0.1 \, m \cdot s^{-1}$



Fig. 2.19: Nu distribution on the top and bottom of flux layer (flux a)



Fig. 2.20: Velocity profile across the domain thickness (centerline; flux a)



Fig. 2.21: Velocity profile across the domain thickness (centerline; flux b)



Fig. 2.22: Velocity profile across the domain thickness (centerline; flux c)



Fig. 2.23: Temperature profile across the domain thickness (centerline; flux a)



Fig. 2.24: Temperature profile across the domain thickness (centerline; flux b)



Fig. 2.25: Temperature profile across the domain thickness (centerline; flux c)



Fig. 2.26: Viscosity profile across the domain thickness (centerline; flux a)



Fig. 2.27: Viscosity profile across the domain thickness (centerline; flux b)



Fig. 2.28: Viscosity profile across the domain thickness (centerline; flux c)



Fig. 2.29: Effect of flux viscosity on the velocity profile $(U_b = 0.06 \, m \cdot s^{-1})$



Fig. 2.30: Effect of flux viscosity on the velocity profile (interfacial shear stress 5 $N \cdot m^{-2}$)



Fig. 2.31: Relationship between interfacial shear stress and flux/steel velocity



Fig. 2.32: Nu number as a function of bottom shear velocity



Fig. 2.33: Nu number as a function of steel velocity

with temperature dependent viscosity			
Domain Length, $L(m)$	10		
Domain Thickness, $H(m)$	1		
Density, $\rho \ (kg \cdot m^{-3})$	1		
Reference Molecular Viscosity, $\mu_0 \ (Pa \cdot s)$	0.1		
Thermal Conductivity, $k \ (W \cdot m^{-1} \cdot \mathbf{K}^{-1})$	0.1		
Specific Heat, $C_p (J \cdot kg^{-1} \cdot m^{-1})$	1000		
Constant for viscosity evaluation, a (K ⁻¹)	0.05		
Drag velocity, $U(m \cdot s^{-1})$	1.414		
Top temperature, T_1 (°C)	100		
Bottom temperature, T_0 (°C)	0		

 Table 2.1: Parameters for simulation of drag flow

 with temperature dependent viscosity

Table 2.2: Parameters for simulation ofbuoyant convection in a thin layer

Domain Length, $L(m)$	0.2
Domain Thickness, $H(m)$	0.01
Density, $\rho \ (kg \cdot m^{-3})$	2500
Molecular Viscosity, μ (Pa·s)	0.1
Thermal Conductivity, $k \ (W \cdot m^{-1} \cdot \mathbf{K}^{-1})$	3
Specific Heat, $C_p (J \cdot kg^{-1} \cdot m^{-1})$	2000
Thermal Expansion Coefficient, β (K^{-1})	2.4×10^{-5}
Left wall temperature, T_c (°C)	0
Right wall temperature, T_h (°C)	100

Case1	Case2
10	0.111125
1	$0.0054 \sim 0.01465$
1000	960.3
0.178	0.0195
0.1	0.14154
1000	1453.2
0.001453	0.00107
5	5
9.8	9.8
1780	200.21
4000	$4000\sim80000$
	Case1 10 1 1000 0.178 0.1 1000 0.001453 5 9.8 1780 4000

Table 2.3: Parameters for validation cases against experiments

Table 2.4: Parameters for variable viscosity and thickness simulations

Domain Length, $L(m)$	0.7
Domain Thickness, $H(m)$	0.01
Density, $\rho \ (kg \cdot m^{-3})$	2500
Molecular Viscosity, $\mu \ (Pa \cdot s)$	f(T)
Thermal Conductivity, $k \ (W \cdot m^{-1} \cdot \mathbf{K}^{-1})$	3
Specific Heat, $C_p (J \cdot kg^{-1} \cdot m^{-1})$	2000
Thermal Expansion Coefficient, β (K^{-1})	$2.4\!\times\!10^{-5}$
Top temperature, T_c (°C)	1000
Bottom temperature, T_h (°C)	1550

	Flux (a)	Flux(b)	Flux (c)
Equation	2.15	2.15	2.14
B (K)	-	-	$23,\!880$
Reference molecular Viscosity, $\mu_0 \ (Pa \cdot s)$	0.87	0.39	0.09
Reference temperature, T_0 (°C)	1300	1300	1550
Solidification temperature, T_s (°C)	850	1125	-
n	3.2	1.3	-
Top temperature, T_c (°C)	850	1125	1000
Bottom temperature, T_h (°C)	1550	1550	1550

Table 2.5: Parameters for shear velocity simulations

Chapter 3. Large Eddy Simulation of Heat Transfer by a Circular Impinging Jet

3.1 Introduction

Impinging jets are encountered in numerous industrial applications. Due to their high heat and mass transfer characteristics, impinging jets are efficient in heating, cooling and drying surfaces as well as in deposition of thin films. Specific applications include cooling of gas turbines,^[24] cooling of electronic devices,^[25, 26] processing of metal and glass, removing particles from a surface^[27] and drying of films and papers.^[28] Beside these applications which take advantage of the transport characteristics, impinging jets also used by vertical take off and landing (VTOL) aircraft,^[29, 30] for chemical vapor deposition of electrochromic glass,^[31] and even to enhance entanglement in non-woven fabrics.^[32]

Figure 3.1 shows a schematic of a circular impinging jet. The geometry is characterized by the jet diameter D and the nozzle to plate distance H. The variables can be made non-dimensional by using the jet diameter and jet inlet velocity as characteristic length and velocity scales. The jet Reynolds number and the fluid Prandtl number are the two key parameters characterizing the rates of heat and mass transfer from or to the jet. The flow structure of the impinging jet can be divided into three distinctive regions: the free jet region, the stagnation region and the wall jet region. In the free jet region, the jet entrains ambient fluid that results in the expansion of the jet and the corresponding changes in the temperature distribution. In the stagnation region, the flow is turned in the radial direction. In the wall jet region, the flow is characterized by a strong radial outflow. The jet slows down in the radial direction as it exits the flow domain. It is observed that the level of turbulence in the jet has a significant effect on the heat transfer rate to the impingement surface. One particular process involving jet impingement which is of direct interest to us is continuous casting of steel.^[14] In a continuous caster of steel, liquid steel flows from a tundish into a mold through a submerged entry nozzle. A jet of fluid with a complex velocity distribution exits the nozzle and impinges on a narrow face of the mold which is cooled by a water flow. The flow in the mold, and heat transfer to the solidifying shell, are determined primarily by the dynamics of this confined obliquely impinging jet. For the steel industry, it is important to predict the rate of solidification and the instantaneous turbulence fields in order to understand the origin of defects in the cast steel. This rate of heat transfer also is needed in estimating the cooling loads required to operate the plant.

Because of the difficulties in accurately measuring the temperature fields inside the caster, computations are the only way to estimate such heat loads and to predict local solidification processes. Previous studies reviewed below have shown the difficulties of accurately predicting impinging jet heat transfer with the Reynolds averaged approach in conjunction with turbulence models. The difficulties arise primarily in characterizing the turbulence in the impinging region where the heat transfer rates can be very large. In our ongoing research, we are developing Large Eddy Simulation (LES) models of the entire casting process including the flow through the nozzle. However, before such predictions are relied upon, experience on the accuracy and grid resolution of the LES approach for simplified configurations is needed. In the present work, we have computed the rates of heat transfer of a circular impinging jet on a planar surface and compared the heat transfer rates with experimental data. In this work we present the time dependent and time-averaged flow and temperature fields for three Reynolds numbers with high grid resolutions. It is observed that the LES approach, while expensive in computational effort, gives accurate predictions of the heat transfer rates.

3.2 Literature Review

There are numerous experimental studies on the impinging jet heat transfer in the literature. Extensive reviews on impinging jet heat transfer exist in the literature. Jambunathan et al.^[33] examined a wide range of experimental data for the heat transfer rate from single turbulent circular impinging jet and proposed a correlation based on the reviewed data. Viskanta^[34] performed an comprehensive review of the heat transfer characteristics of single and multiple isothermal turbulent air and flame impinging jets. The review discussed various aspect of the jet impingement problem and the effect of different impinging jet configurations. It also pointed out the difficulties in comparing different experimental measurements due to numerous differences such as nozzle design, turbulence in the jet, measurement technique, etc.

Experimental studies reveal many features of impinging jet heat transfer. The work of Huang and El-Genk^[35] showed that the maximum stagnation heat transfer occurred at jet to plate spacing of 4.7. Elison and Webb^[36] reported that the Nu correlates approximately with $Re^{0.5}$ and $Re^{0.8}$ for turbulent and laminar jets, respectively. They also showed that the Nu is independent of jet to plate distance for laminar jets. If the jet has a different temperature than the ambient, then its entrainment effect can be described by an effectiveness.^[37, 38] If the jet impinges at an angle to the surface, then the point of maximum heat transfer will shift away from the geometrical impingement point towards the compression side of the jet.^[39] The displacement is primarily the function of impingement angle.^[40] The maximum heat transfer rate decreases with increasing jet inclination.^[41]

Different configurations of the impinging jet also affect the heat transfer to the impingement plate. Multi-channel impinging jets deliver much higher heat transfer rate than a conventional jet.^[42] Swirl impinging jets were found to increase the heat transfer rate as well as significantly enhance uniform radial distribution of heat transfer rate.^[42] Confinement makes local heat transfer coefficients more sensitive to

Reynolds number and jet to plate distance.^[43] Confined impinging jets are also less efficient with respect to heat transfer rate.^[44] The exit condition of the nozzle and the nozzle geometry also affect the heat transfer rate.^[45–47]

Several experiments were dedicated to the identification of structures in the impinging jet and their effects on the heat transfer. Sakakibara et al.^[48,49] used digital particle image velocimetry (DPIV) and laser induced fluorescence to study structure of a plane impinging jet. They found that the counter rotating vortex pair in the stagnation region contributes significantly to the heat flux. The work of Hwang et al.^[50] showed that the flow structures of the jets are affected strongly by the nozzle exit conditions and vortices generated around the jet periphery and hence affect the heat transfer characteristics of the impingement surface.

The different configuration of the jet inlet conditions, the ambiguity of the boundary condition at the impingement plate (between constant temperature and constant heat flux for most expeirments) and the indirect measurement of heat flux all make it difficult to compare numerical simulations directly with experimental data. Baughn and Shimizu^[51] performed experiments with well controlled fully developed jet profile and uniform heat flux thermal boundary condition. Cooper et al.^[52] perform measurements of a turbulent jet impinging with a fully developed pipe inlet. Mean velocity profile near the plate and the three Reynolds stress components in the axial-radial plane were reported. Hollworth and Gero^[38] performed experiments by measuring the heat flux directly and keeping the target plate at constant temperature. All of these works are suitable for numerical comparison. The data from Hollworth and Gero^[38] was used to compare with the simulation results in this work.

There are several efforts in the literature which attempted to predict the turbulent impinging jet heat transfer accurately using the Reynolds-averaged Navier-Stokes (RANS) approach. Due to the limitation of the method, the RANS method can not predict the transient characteristic of the turbulent impinging jet. Further more, the standard k- ε model and the Reynolds stress model used in many commercial packages failed to predict the correct average heat transfer rate in the impingement region.^[53-56] The successful RANS models in the literature are all modified models which address the turbulent phenomena in the impingement region using special practices. Among those models are the normal velocity relaxation turbulence model (V2F) proposed by Behnia and Parneix,^[54,57] the modified k- ε - f_{μ} model proposed by Park and Sung,^[56] the second-order, single point closure model proposed by Dianat and Fair-weather,^[55,58] etc.

There are not many numerical studies of the impinging jet that utilized Direct Numerical Simulation (DNS) or LES. Most of those were limited to low Reynolds numbers. Laschefski et al.^[59] performed DNS simulations of the flow field of impinging axial and radial slot jets for low Reynolds numbers. The transition of the impinging jet from steady to periodic and then to chaotic was studied. Chung et al.^[60] performed 2-D DNS simulations of unsteady impinging jet. The Reynolds numbers studied were under 1,000. It is found that the amplitude of the oscillation of the heat transfer at the stagnation region is as high as 20% of the mean value. They also showed that the vortex location has much stronger effect on Nu at the stagnation point than the strength of the vortex.

Voke, Gao and Leslie^[61, 62] performed LES simulation to study the flow and thermal fields of plane impinging jets with Reynolds number 6,500. It was reported that the principle mechanism for the generation of large-scale temperature fluctuations to be the instability at the edge of the jet. Olsson and Fuchs^[63] applied LES to study a forced semi-confined circular impinging jet with jet Reynolds number of 10⁴. The effects of spatial resolution and SGS models were studied. Their results showed that the SGS models have little influence on the velocity field solution, but the effect is more pronounced for turbulence statistics. The dynamics of vortices were also studied and they found that the primary vortices have helix structure instead of being axisymmetric. Cziesla et al.^[64] used LES in the study of heat transfer form an array of slot impinging jets with Reynolds number under 3000. They investigated the performance of a LES using a logarithmic wall law, which was found to be fairly good for the Reynolds number range of 600 to 3000.

3.3 Numerical Method

In this work, we carried out the computation of the turbulent flow and heat transfer of circular impinging jets by numerically integrating the three-dimensional unsteady Navier-Stokes equations in cylindrical coordinates. Fig. 3.2 shows a schematic of the simulation domain and the coordinate system used in this study. Three jet Reynolds numbers were studied, 5000, 10000 and 20000. The distance between the nozzle exit and the impingement plane is fixed at 5 jet diameters. For all the simulations, a static $k \mod l^{[65]}$ was used as the sub-grid scale model. To suppress unphysical overshoots and undershoots originated from the dispersive error of the numerical scheme, a multi-dimensional flux limiter^[66] is applied for all the simulations.

3.3.1 Governing equations

This work applies Large Eddy Simulation in which the large scale motions are computed explicitly and the small scale (subgrid-scale) motion is modelled. In LES, a filter operation is defined to decompose the velocity u_i into the sum of a filtered component \bar{u}_i and a SGS component u'_i .^[67] The filtered velocity \bar{u}_i is three dimensional and time dependent. This work uses a grid based filtering procedure where the filtered velocity \bar{u}_i represents the velocity u_i averaged over a grid cell.

The time-dependent Navier-Stokes equations are solved for the filtered velocities. The filtered Navier-Stokes equations can be written in index summation form where repeated indices translate to summation over the three coordinate directions. Mass continuity:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{3.1}$$

Momentum:

$$\rho_0 \left(\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} \right) = -\frac{\partial \bar{p}}{\partial x_j} + \mu \frac{\partial}{\partial x_j} \left(\frac{\partial \bar{u}_i}{\partial x_j} \right) + \frac{\partial Q_{ij}}{\partial x_j} + \delta_{i3} \rho_0 \beta (\bar{T} - T_0) g \qquad (3.2)$$

Energy:

$$\frac{\partial \bar{T}}{\partial t} + \frac{\partial (\bar{u}_i \bar{T})}{\partial x_i} = \frac{k}{\rho_0 C_P} \frac{\partial}{x_i} \left(\frac{\partial \bar{T}}{\partial x_i}\right) + \frac{\partial Q_{Ti}}{\partial x_i}$$
(3.3)

where \bar{u}_i is filtered velocity. \bar{p} is filtered pressure. \bar{T} is filtered temperature. μ is the molecular viscosity. ρ is the density given at a reference temperature T_0 . β is the volumetric thermal expansion coefficient. g is the gravity acceleration. k is the thermal conductivity. C_p is the heat capacity. The momentum equations and the energy equation are coupled through a buoyancy force term based on Boussinesq approximation.^[11] This term is only nonzero in the z direction, as implied by δ_{i3} in Eq. 3.2. The buoyancy term is only switched on in the simulations of continuous casting and ignored during all the simulations of impinging jet.

The term Q_{ij} represents the sub-grid momentum fluxes and can be expressed as:

$$Q_{ij} = \bar{u}_i \bar{u}_j - \overline{u_i u_j} \tag{3.4}$$

The trace-free part of the sub-grid flux is modelled in terms of the resolved scales. Models based on eddy viscosity are frequently used, in which the trace free part τ_{ij} is assumed to be proportional to the symmetric strain rate tensor of the resolved scales. The proportionality constant μ_T is defined by:

$$\tau_{ij} = Q_{ij} - \frac{1}{3}Q_{kk} = 2\mu_T \bar{S}_{ij} \tag{3.5}$$
$$\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)$$
(3.6)

In LES, the eddy viscosity μ_T is calculated from an empirical correlation with a length scale and a sub-grid velocity scale. The turbulent eddy viscosity model used in this work is one proposed by Horiuti.^[65]

$$\mu_T = C_v K_G^{1/2} \Delta \tag{3.7}$$

where C_v is a constant equals to 0.05, and Δ is the grid length scale given by $\Delta = (\Delta_x \Delta_y \Delta_z)^{1/3}$ (Δ_x , Δ_y and Δ_z are grid sizes in x, y, z directions, respectively). K_G is the sub-grid scale turbulent energy $\overline{u'_i u'_i}/2$. K_G is determined by solving another transport equation given by:

$$\frac{\partial K_G}{\partial t} + \bar{u}_j \frac{\partial K_G}{\partial x_j} = \frac{1}{2} \nu_T \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)^2 - C_{\varepsilon} \frac{K_G^{3/2}}{\Delta} + \frac{\partial}{\partial x_i} \left[\left(\nu_T + C_{kk} K_G^{1/2} \Delta \right) \frac{\partial K_G}{\partial x_i} \right]$$
(3.8)

where $\nu_T = \mu_T / \rho$ is the kinematic turbulent viscosity. C_{ε} is a constant equals to 1.0. C_{kk} is a constant of value 0.1.

The term $Q_T i$ in the energy equation represents the sub-grid heat fluxes and can be expressed as:

$$Q_{Ti} = \bar{T}\bar{u}_i - \overline{Tu_i} \tag{3.9}$$

The sub-grid heat flux is modelled as an extra diffusion terms using the eddy viscosity μ_T from momentum equation and the turbulent Prandtl number Pr_T :

$$\frac{\partial Q_{Ti}}{\partial x_i} = \frac{\mu_T}{\Pr_T} \frac{\partial}{\partial x_i} \frac{\partial \bar{T}}{\partial x_i}$$
(3.10)

The turbulent Prandtl number used in the present simulation is 0.9.

3.3.2 Numerical method

A finite volume method was used to solve equations 3.1 to 3.3. Central difference with second order accuracy was used to discretize the equations on a collocated grid with variables defined at the cell centers. The time integration of the equations was done using a semi-implicit, fractional step method with diffusion terms treated implicitly by the Crank-Nicolson method. The convective and source terms from SGS stresses are advanced explicitly using the second-order Adams-Bashforth method. After applying the semi-implicit procedure, the momentum equations take the following form:

$$\frac{\bar{u}_i^{n+1} - \bar{u}_i^n}{\Delta t} = \frac{3}{2}H_i^n - \frac{1}{2}H_i^{n-1} + \frac{1}{2}\frac{\partial}{\partial x_j}\left[\left(\nu + \nu_T\right)\left(\frac{\partial\bar{u}_i^n}{\partial x_j} + \frac{\partial\bar{u}_i^{n+1}}{\partial x_j}\right)\right] - \frac{\partial\bar{P}^{n+1}}{\partial x_i} \quad (3.11)$$

where H_i is given by

$$H_i = -\frac{\partial}{\partial x_j} \left(\bar{u}_i \bar{u}_j \right) + \frac{\partial}{\partial x_j} \left(\nu_T \frac{\partial \bar{u}_j}{\partial x_i} \right)$$
(3.12)

In the fractional step method an intermediate velocity field ($\bar{\tilde{u}}_i$) is calculated by neglecting the pressure gradient term in the momentum equations.

$$\frac{\bar{\tilde{u}}_i - \bar{u}_i^n}{\Delta t} = \frac{3}{2} H_i^n - \frac{1}{2} H_i^{n-1} + \frac{1}{2} \frac{\partial}{\partial x_j} \left[\left(\nu + \nu_T\right) \left(\frac{\partial \bar{u}_i^n}{\partial x_j} + \frac{\partial \bar{\tilde{u}}_i}{\partial x_j}\right) \right]$$
(3.13)

Then in the second half-step the $\bar{\tilde{u}}_i$ field is corrected to satisfy continuity by solving for the pressure field at the next time step. Subtracting Eq. 3.13 from Eq. 3.11 gives the following expression:

$$\frac{\bar{u}_i^{n+1} - \bar{\tilde{u}}_i}{\Delta t} = \frac{1}{2} \frac{\partial}{\partial x_j} \left[(\nu + \nu_T) \frac{\partial (\bar{u}_i^{n+1} - \bar{\tilde{u}}_i)}{\partial x_j} \right] - \frac{\partial \bar{p}^{n+1}}{\partial x_i}$$
(3.14)

If we express the right hand side of the above equation in terms of the gradient of a scalar value Φ , we can get the following expression:

$$\frac{\bar{u}_i^{n+1} - \bar{\bar{u}}_i}{\Delta t} = -\frac{\partial \Phi^{n+1}}{\partial x_i} \tag{3.15}$$

A Poisson equation can be obtained by applying the divergence operator to the above equation:

$$\frac{\partial}{x_i} \left(\frac{\partial \Phi^{n+1}}{x_i} \right) = \frac{1}{\Delta t} \left(\frac{\partial \bar{\tilde{u}}_i}{\partial x_i} - \frac{\bar{u}_i^{n+1}}{\partial x_i} \right) = \frac{1}{\Delta t} \frac{\partial \bar{\tilde{u}}_i}{\partial x_i}$$
(3.16)

Notice that the velocity field should satisfy the continuity equation thus divergence free, so the term containing velocity vanishes in the above equation. The pressure and the scalar Φ are related by the following equation:

$$\bar{p}^{n+1} = \Phi^{n+1} - \frac{\Delta t}{2} \left(\nu + \nu_T\right) \frac{\partial}{x_i} \left(\frac{\partial \Phi^{n+1}}{x_i}\right)$$
(3.17)

3.3.3 Initial and boundary conditions

Since the scheme involves time integration of a turbulent flow field the initial conditions will not affect the accuracy of the final solution, although it will influence the integration time to reach a statistically stationary state. Zero flow velocities and the ambient temperature are assumed initially throughout the domain.

The simulation domain and boundary conditions are shown in Fig. 3.2. The jet comes out of a long nozzle and enters a large domain. After impinging on the plate, the jet turns in radial direction and exits from the openings at the side. As the flow in the nozzle is not the main concern of this study, the inlet condition used for the nozzle is a uniform flow. The temperature of the inlet is set to a constant value equals the ambient air temperature. The length of the nozzle is long enough for the flow to become fully developed when exiting the nozzle and becomes a free jet. The nozzle wall is modelled as adiabatic as the temperature of the air equals the ambient temperature. The impingement plate is set to a constant temperature the same as in the experiment. The top and wide walls of the tank is modelled as adiabatic walls. It is assumed that the domain is large enough that the boundary of the tank will not have a significant effect on the flow of the impinging jet. The opening for air to exit is prescribed a convective boundary condition with convective velocity equals the average exit velocity.

3.3.4 Computational details

Table 3.1 lists the parameters and material properties used in the simulation. The dimensions of the domain are the same as in the experiment performed by Hollworth and Gero.^[38] The operating conditions are the same conditions as the experiments. The material properties for air are standard values at one atmospheric pressure and room temperature.

One representative grid used in the simulation for Reynolds number 10,000 is shown in Fig. 3.3. Detailed grid information for all the cases is documented in Table 3.2. For all simulations, the grid in the circumferential direction was uniform. In the other two directions, the grids were stretched using a ratio below 1.03. To resolve the large velocity and temperature gradients near the impingement surface and at the shear layer of the jet, the grid space is the smallest in those regions. The minimum grid spacing in axial and radial direction for all three cases is listed in Table 3.2.

The FORTRAN computer program UIFLOW developed in the CFD lab at the University of Illinois at Urbana Champaign was used for the computation. Time steps satisfying the Courant-Friedrichs-Lewy $(CFL)^{[10]}$ stability condition for the convective terms and diffusion time step were used. The computations were performed on 1.7 GHz personal computers with 2 GB of memory, requiring ~20 (CPU) seconds per time step.

3.4 Results and Discussions

The instantaneous velocity and temperature fields for the case with Reynolds number 10,000 are shown in Fig. 3.4 and Fig. 3.5. The jet entrains surrounding fluid and expands as it travels downward. The shear layer between the jet and the ambient air is unstable and generates several small eddies. After the jet impinges on the plate, the flow is diverted in the radial direction and becomes a wall jet. The wall jet fluctuates frequently and shows a wavy pattern instantaneously. In most of the domain, the temperature remains at the ambient temperature (25 °C). The eddies near the plate entrain lower temperature fluid and carry it upwards. The intermittent nature of the wall jet makes the edge of cooled fluid layer very zagged. The instantaneous velocity and temperature fields for other Reynolds numbers are similar in nature.

Figure 3.6 and Fig. 3.7 shows the mean velocity and temperature fields of the impinging jet simulation for Reynolds number 10,000. The fields were averaged first in time and then averaged in space over the homogeneous azimuthal direction. After the jet discharges into the ambient air, it entrains air along the way and becomes wider in diameter. When the jet impinges on the plate, it forms a stagnation region and the flow is diverted in the radial direction. The radial flow, which is also referred to as wall jet, first increases in speed as it gets away from the stagnation point. As it goes further out and spreads in the radial direction, the radial velocity of the wall jet decreases with increasing radial distance from the stagnation point. The thickness of the wall jet also increases as the radial distance increases. This can be seen clearly in the radial velocity profiles at different distances from the axis (Fig. 3.9). The temperature in most of the domain remains very close to the ambient temperature of the air (25 °C). It is only near the plate where the air is being cooled that large temperature gradients are present. The temperature gradients are the largest in the stagnation region.

increases. The distortion in temperature field near the end of the domain is due to the effect of outlet.

In Fig. 3.8, the radial velocity profiles are compared with available experimental data at r/D = 1. Two sets of experimental data are used. The data from Landreth and Adrian^[68] are Particle Image Velocimetry (PIV) measurements of a impinging jet with Reynolds number of 6,564 and jet to plate distance of 4D. The other set of data is from Didden and Ho,^[69] with the jet Reynolds number at 19,000 and the jet to plate distance at 4D. To make the velocity profile comparable between different Reynolds numbers, the velocity is normalized by the bulk velocity of the jet. With a larger jet to plate distance, the jet is expected to be wider and the radial flow to be weaker. Compared with experimental data, the radial velocity is higher away from the plate and lower when close to the plate. The comparison results show the solution of velocity fields is fairly accurate. When normalized, there is little difference between the mean profiles of difference Reynolds numbers, so the plotted mean velocity field in Fig. 3.6 and Fig. 3.9 is representative for all Reynolds numbers. As for mean temperature fields, all cases are similar except that the temperature gradients near the plate are higher for larger Reynolds numbers.

The profiles of RMS statistics for the case with Reynolds number 20,000 are shown in Fig. 3.10 to Fig. 3.13. The RMS velocities are normalized by the bulk velocity of the jet. In the impinging jet, The axial RMS velocity remains almost constant and then decreases to zero rapidly as it gets closer than 0.1D to the plate . In the wall jet, the peak value of RMS axial velocity decrease as the wall jet moves away from the center. The peak position shifts away from the plate as well. The RMS radial velocity first increases and then decreases as the wall jet develops. The RMS radial velocity profiles share many common features as measured profiles by Cooper et al.^[52] They are also comparable quantitatively, though the simulation differs from the experiments in Reynolds number and jet to plate spacing. The RMS azimuthal velocity profiles have the similar shape as the RMS radial velocity profiles, but the value keeps decreasing as the wall jet develops. The RMS temperature profile peak shifts away from the plate as the radial distance increases, but the peak value has little change. The effect of Reynolds number on the RMS statistics is shown in Fig. 3.14. Profiles at r/D = 2.0D are compared for different Reynolds numbers. Decreasing Reynolds number has little effect on the axial RMS velocity, but it decreases the radial and azimuthal velocity slightly and moves the RMS temperature peak away from the impingement plate.

The instantaneous Nu in the plate is plotted in Fig. 3.15. The picture shows contours which look like broken concentric rings. This instantaneous picture indicates the fluctuations of heat flux in the radial as well as in the circumferential directions. The average and RMS Nu number profiles in the radial direction are shown in Fig. 3.16, Fig. 3.17 and Fig. 3.18 for simulation with Reynolds number 5,000, 10,000 and 20,000. The model is able to capture the higher heat transfer peaks in the stagnation region. The RMS Nu is quite large for all the simulations, with its maximum value reaching around 50% of the maximum mean Nu. The predicted Nu beyond radial distance of 6D is lower than the experimental value. It decreases first and then increases to approach the experimental value near r/D = 8. This discrepancy is due to the influence of the outlet of the domain. The peak in the Nu of the simulation results is somewhat larger than the measurement in the stagnation region. The cause of this may most likely be due to the different jet inlet profiles. In the simulation, the jet is discharged from a long pipe, while in the experiments, the jet is discharged from an orifice in a plenum.^[38] The difference in the inlet condition may contribute to the larger Nu predicted by the simulation. The agreement between the simulation results and experiments is the best for the case of jet Reynolds number 5,000 because this case has a relatively finer grid with respect for the given jet Reynolds number. Overall, the numerical results match the experiment data of Hollworth and $\text{Gero}^{[38]}$

reasonably well, the maximum difference between the predicted and the experimental value being around 20%.

3.5 Summary

The LES model produces satisfactory results for heat transfer of circular impinging jet. The predicted mean and RMS velocity achieve reasonable agreement with the experimental data in the literature. The model captures the higher heat transfer rate to the impingement plate, the maximum difference between predicted value and measurements being around 20%. Thus the current numerical method is validated for application to the continuous casting of steel.

3.6 Figures and Tables



Fig. 3.1: Schematic of an impinging jet



Fig. 3.2: Simulation domain and boundary conditions



Fig. 3.3: Grid used in simulation for Reynolds number of 10^5



Fig. 3.4: Instantaneous velocity field $(Re=10^5)$



Fig. 3.5: Instantaneous temperature field $(Re=10^5)$







Fig. 3.7: Mean temperature field $(Re=10^5)$



Fig. 3.8: Radial mean velocity profile at r=1.0D, experimental data (a) from Landreth and Adrian,^[68] experimental data (b) from Didden and Ho^[69]



Fig. 3.9: Development of radial mean velocity profile $(Re=10^5)$



Fig. 3.10: RMS axial velocity profiles $(Re=2\times10^5)$



Fig. 3.11: RMS radial velocity profiles $(Re{=}2{\times}10^5)$



Fig. 3.12: RMS azimuthal velocity profiles $(Re=2\times10^5)$



Fig. 3.13: RMS temperature profiles $(Re=2\times10^5)$



Fig. 3.14: Effect of Reynolds number on RMS statistics



Fig. 3.15: Instantaneous Nu distribution on the impingement plate ($Re=10^5$)



Fig. 3.16: Radial distribution of mean and RMS Nu for $Re=5\times10^4$, experimental data from Hollworth and Gero^[38]



Fig. 3.17: Radial distribution of mean and RMS Nu for $Re=10^5$, experimental data from Hollworth and Gero^[38]



Fig. 3.18: Radial distribution of mean and RMS Nu for $Re=2\times10^5$, experimental data from Hollworth and Gero^[38]

Inlet diameter D (mm)	10		
Nozzle to plate distance $H(mm)$	50		
Inlet temperature T_p (°C)	25		
Ambient temperature T_a (°C)	25		
Impingement surface temperature T_s (°C)	8		
Density of air $\rho \ (kg \cdot m^{-3})$	1.2		
Molecule viscosity of air $m (Pa \cdot s)$	17.85×10^{-6}		
Thermal conductivity $k \ (W \cdot m^{-1} \cdot \mathbf{K}^{-1})$	0.025		
Specific heat of air $C_p (J \cdot kg^{-1} \cdot \mathbf{K}^{-1})$	1006		
Prandtl number Pr	0.71		
Renolds number Re	5000	10000	20000
Inlet bulk velocity $(m \cdot s^{-1})$	7.4375	14.875	29.75
Time step Δt (s)	2×10^{-6}	$1\!\times\!10^{-6}$	5×10^{-7}

Table 3.1: Parameters for impinging jet simulation

Table 3.2: Grid details for simulations

Case Re	$5,\!000$	10,000	20,000	
Grids in the nozzle		$65 \times 20 \times 64$		
Grids in the tank	$80\times105\times64$	$95\times105\times64$	$110\times105\times64$	
Minimum Δx	0.015D	0.01D	0.006D	
Δy (radial direction)	$0.02 \sim 0.237 D$			
Total number of grids	614,000	721,000	822,400	

Chapter 4. Large Eddy Simulation of Transient Flow and Heat Transfer in the Mold Region

4.1 Introduction

This section focuses on the turbulent flow and heat transfer in the mold region of a continuous caster of steel. Continuous casting of steel is a very complex process which involves turbulent fluid flow, heat transfer, multi-phase flow and solidification. The flow of molten steel and the associated transport of superheat in the upper portion of the liquid steel pool is critical to the quality of the final product. The molten steel, driven by gravity, flows from the tundish into a submerged entry nozzle and then goes into the mold cavity through two or three bifurcated ports near the bottom of the nozzle. The flow out of the nozzle forms steel jets which travel across the molten pool in a confined space formed by the solidifying steel shell. The jets then impinge obliquely onto the narrow face, causing locally high heat transfer rate to the shell. The impingement point often coincides with the exit of the mold, where the solidified shell must be thick enough to withstand the ferrostatic pressure to prevent molten steel from bursting through the shell to cause an expensive "breakout". Many casting operations restrict the casting speed according to the superheat in order to minimize the danger of such breakouts.

From the impingement point, the jets split upward and downward, flowing to create an "upper roll" above each jet and "lower roll" in the lower regions of the strand. The exact nature of the flow pattern depends on the nozzle port shape and angle, the submergence depth, the cast section size, the injected gas fraction, and the extent of electromagnetic stirring, and is studied elsewhere.^[70–73]

At the top surface, the molten steel should retain sufficient superheat and speed to avoid solidification of the meniscus, which can lead to subsurface "hook" formation and associated defects in the solidified product.^[14] Insufficient superheat thus leads to increased surface defects. Finally, the superheat in the mold controls the internal microstructure and the associated macro-segregation of the final product. Specifically, the degree of superheat controls the formation and remelting of crystal nuclei that grow into the equiaxed grains that eventually comprise the center of the strand. These grains are beneficial for avoiding centerline segregation. Excessive superheat in the mold is thus associated with larger columnar-grains and increased segregation and internal cracking problems. To avoid these defects, the liquid flow pattern and superheat must be carefully optimized.^[14]

Quantifying the turbulent heat transfer in the mold region of continuous casters is of great importance to the understanding of the process and defect control. It is especially important to quantify the peak heat flux that causes thinning of the shell around the impingement region centered at the narrow face, and the amount of superheat delivered to the shell during the critical early stages of solidification at the meniscus. However, the harsh high temperature environment of the process makes it very difficult for direct measurement of the flow and temperature field. Only a few measurements in the molten steel caster have been attempted.^[74–76] Scaled or full dimension water models have provides most of the important insights into the transient flow features of the process,^[75,77–80] but are severely limited in studying heat transfer. Thus, previous understanding of transient flow and heat transfer in the molten steel pool has relied heavily on computational models.^[71,81]

This work uses Large Eddy Simulation to calculate transient flow and heat transfer in the liquid pool of a typical continuous caster of thin steel slabs. The numerical method used is the same as that used in the study of circular impinging jet heat transfer and is validated through comparison with laboratory measurements in the literature. The caster simulation results are then compared with temperature measurements obtained by inserting thermocouple into an operating thin slab caster.

4.2 Review of Literature

Only a few previous studies have simulated turbulent fluid flow and heat transfer within the liquid pool of a continuous casting machine. Thomas and Najjar^[82] developed a two-dimensional finite-element model with the commercial code FIDAP using the two equation k- ε turbulence model. Various solution strategies, relaxation factors and meshes were investigated to help provide guidelines for achieving convergence. The predicted flow patterns and velocity fields showed reasonable agreement with measurements in a water model, and the predicted heat flux profile was consistent with previous measurements. The results were sensitive to the choice of k and ε inlet conditions, wall laws and turbulent Prandtl number. Huang, Thomas and Najjar^[81] used a 3-D steady state, k- ε model to simulate 1/4 of a mold, with the SIMPLE finite-volume solution method. The calculated temperatures agreed with one of the few published measurements, by Offerman.^[74] Over half of the superheat was shown to be removed in the mold and the maximum heat input to the shell occurs near the jet impingement point on the narrow face. Furthermore, this heat input increases directly with superheat temperature difference and casting speed.

Seyedein and Hasan^[83,84] performed a 3-D finite-difference simulation on a staggered grid with a low Reynolds number k- ε model, including the liquid, mushy, and solid regions within a stainless steel caster. Different heat extraction rates from the solid surface boundaries were studied. Creech^[73] evaluated various Reynolds-average Navier-Stokes (RANS) models, including the standard and low-Re models, using the finite-difference package CFX.^[85] Differences in peak heat flux up to 240% were found between different turbulence models, wall laws, and grid size near the shell. The low-*Re* model in particular over-predicted peak heat flux if the grid was too coarse. Thomas et al.^[86] compared the best of these models with temperature measurements in a stainless-steel thin slab caster^[87] with reasonable matching. A corresponding heat transfer model of the solidifying shell and mold successfully matched mold temperature, cooling water heat-up, and shell thickness measurements.

These previous models all employed popular RANS methods to model the turbulence. RANS methods decompose variables into mean and fluctuating components and feature separate transport equations for variables such as kinetic energy and dissipation, that are based on empirical constants. They also use special empirical boundary conditions called "wall laws". These methods are popular because they can be performed on course grids using few computational resources. They have proven successful for flow problems,^[71, 73, 83] as documented previously.^[72, 76] However, the accuracy of the corresponding RANS heat transfer models, with their associated wall laws, have received much less attention, especially in flowing metal systems. Furthermore, the RANS models are not well suited for the quantitative prediction of transient phenomena. Other numerical methods exist to simulate turbulent fluid flow and the associated heat transfer.

The most accurate method is Direct Numerical Simulation (DNS), which resolves the details of turbulent flow by solving the Navier-Stokes equations on a very fine computational grid. Because the grid size needed to resolve all the turbulence scales increases with the $9/4^{th}$ power of Reynolds number,^[10] fully turbulent processes such as continuous casting cannot be fully resolved using DNS even with the most powerful computers. Large Eddy Simulation (LES) is a method with intermediate complexity between DNS and RANS methods. In LES, the grid is fine enough to resolve the large scales of turbulence while the small scales are assumed to be isotropic and are handled with a sub-grid-scale (SGS) viscosity model. With better efficiency than DNS and a better fundamental basis than RANS, LES is used in the current work to simulate the complex flow and heat transfer phenomena in continuous casting of steel. Initial efforts to apply LES to the continuous casting steel, have successfully matched PIV and other measurements including asymmetric fluid flow and transient particle motion.^[2, 88, 89] Previous efforts have not included transient heat transfer, which is the subject of this work.

Measurements were conducted on the AK Steel thin slab caster in Mansfield, OH, including flow in the mold using a water model, and temperature in the liquid pool of the actual caster. These measurements provide experimental data for the numerical model to compare with. The parallel mold of this caster is $132 \, mm$ thick $\times 984 \, mm$ wide $\times 1.2 \, mm$ long and is described in detail elsewhere.^[87] A stopper rod controls flow through the oval-bored submerged entry nozzle that features three exit ports: two rectangular side ports, and a circular central port at the bottom. The nozzle is submerged 127 mm, measured from the molten-flux steel interface to the top of the side ports.

To study flow in the liquid pool, a full-scale water model was constructed,^[87] including the tundish, the submerged entry nozzle with stopper-rod control, and a 2.6 m long segment of the mold and strand with an automated level control control system. The water exiting from the mold bottom was recirculated back to the tundish, allowing time to establish fully pseudo-steady state conditions. The flow field was visualized by injecting dye into the submerged entry nozzle. Flow velocities were estimated through analysis of successive frames of videotape used to track movement of the die front, knowing the time between frames.^[88] The videotaped flow patterns can thus provide both qualitative and quantitative comparisons with the simulation results.

Measurements of the liquid steel in the mold are difficult owing to the high temperature environment of molten steel and the cramped space between the tundish and mold. The ability of computational models to quantitatively predict fluid velocities has been investigated previously with the help of velocity sensors in an operating steel caster.^[76] For this work, an apparatus was constructed to measure vertical temperature profiles in the liquid steel pool in the mold, as pictured in Fig. 4.1. The apparatus guides a thermocouple down through the top slag layer to a maximum insertion depth into the molten steel of $180 \, mm$. The thermocouple is moved down and up slowly (0.6 mm/s), allowing time for thermal equilibrium at the thermocouple tip. For each data set, temperatures were digitally recorded during both insertion and withdrawal.

An example of the temperature measurements is given in Fig. 4.2. The temperature gradient decreases with increasing submergence depth, owing to the increasing thermal conductivity of the mold powder, liquid flux, and molten steel. The powder/flux and flux/steel interfaces are identified from the different slopes in these three regions. Table 4.1 gives the conditions for five different tests with 129 mm nozzle submergence depth, taken at different distances along the center plane between the wide faces. Two further tests were repeated with 159 mm submergence depth, reported elsewhere.^[90]

4.3 Governing Equations and Numerical Method

In this work, transient flow structures and the corresponding heat transfer is computed by numerically integrating the three-dimensional unsteady Navier-Stokes equations with a Large-Eddy Simulation of the liquid pool region of the AK Mansfield thin slab caster. Fig. 4.3 shows the simulation domain and coordinate system. It features the entire nozzle and one half of the top 1.2 m of the mold and strand region in the liquid pool, assuming symmetry about the center plane. The computational grid features 1.6 million cells, as shown in Fig. 4.4 and Fig. 4.5. Although this grid captures most of the structures that control the flow, it does not resolve the very smallest eddies, so a static $k \mod l^{[65]}$ is used as the sub-grid-scale viscosity model. The grid is sufficiently fine that the SGS model is not needed for stability, so a computation without the SGS k-model is also performed for comparison.

4.3.1 Governing equations

The time-dependent Navier-Stokes and energy equations are solved in a Cartesian coordinate system for the velocity, pressure, and temperature distributions. The governing equations, the formulation of the SGS model and the numerical method in solving the equations are discussed in details in Chapter 3.

4.3.2 Initial and boundary conditions

The boundary conditions are summarized on the simulation domain in Fig. 4.3, which contains one half of the liquid pool of the continuous caster. The left wall represents the symmetry plane between narrow faces where the normal velocity component and the gradients of other variables are thus set to zero. The minor asymmetries ignored by this assumption are investigated elsewhere.^[88,89] The wide and narrow face domain boundaries represent the dendritic solidification front of the inside of the solidifying steel shell. In a real caster, the solidifying shell grows in thickness with distance below the meniscus. However, for the domain length simulated, the maximum shell thickness is less than 4% of the domain width. Thus, the wide and narrow faces were assumed to be straight walls with no-slip boundary conditions, with the axial (z) velocity component set to the casting speed, to match the withdrawal rate of the shell. Neglecting the shell thickness effect, which is also intrinsic to water models, slightly exaggerates flow variations through the domain thickness, as investigated elsewhere.^[88] However, the effect on jet impingement along the narrow face, which is of greatest concern in this work, is minor. For thermal boundary conditions, these walls are set to a constant temperature equal to the liquidus solidification temperature of the steel alloy (1775 K). The domain outlet is artificially cut off at a horizontal plane 1.2 m below the meniscus, where a constant pressure is prescribed. Heat leaves the domain outlet only through advection. Level fluctuations of the top free surface are around 5 mm.^[88] This minor shape change was assumed to have negligible effect on the flow, so the top surface was modelled as a rigid plane with zero normal velocity and zero gradients prescribed for other variables. An adiabatic thermal boundary condition is prescribed over the top surface, because previous work has shown that heat loss through the insulating flux and powder layers is small (see Chapter 2).

Flow through the nozzle is not the main concern of this study, so the inlet condition to the nozzle is a uniform velocity corresponding to the flow rate and measured casting speed. Non-uniformities from a properly-centered stopper rod flow control are expected to be small. The nozzle is long enough for the flow to become fully turbulent before entering the mold. The nozzle inlet temperature is set to a constant "casting temperature" measured in the tundish. The nozzle walls are assumed adiabatic, as the alumina-graphite has good insulation and the residence time is very short.

In transient simulation of a pseudo-steady flow field, the initial conditions do not affect the accuracy of the final solution, but will influence the integration time needed to reach a statistically stationary state. Zero flow velocities and the liquidus solidification temperature of steel are assumed initially throughout the domain.

4.3.3 Computational details

Table 4.2 lists the parameters and material properties used in the simulation. The dimensions and operating conditions are chosen to match the conditions of the experiments conducted on the real steel caster. The material properties for liquid steel are standard values for typical steel. The thermal expansion coefficient is determined using the available liquid density properties.^[91] A value of 1.0×10^{-4} was used. For the small temperature difference in the present problem (57K), the Boussinesq approximation is valid.

A computational grid consisting of 1.64 million cells was used. Curved surfaces are modelled using a stair step grid. The maximum gradients in the flow and the thermal field occur in the jet, its shear layer and near the solidifying shell. Hence the grid was stretched in all directions using a ratio below 1.03 to satisfy the accuracy requirements.

The FORTRAN computer program UIFLOW developed in the CFD lab at the University of Illinois at Urbana Champaign was used for the computation. A time step Δt of 0.0005 s, satisfying the Courant-Friedrichs-Lewy (CFL)^[10] stability condition for the convective terms and diffusion time step was used. The computations were performed on 1.7 GHz personal computers with 2 GB memory, requiring ~32 (CPU) seconds per time step.

4.4 **Results and Discussion**

The discretized Navier-Stokes and energy equations were first integrated in time to reach a statistically stationary state of flow and temperature fields. After the flow reached a fairly stationary state, 15 s of instantaneous flow and temperature fields were computed with a stationary wall conditions at the wide and narrow faces. Then a drag velocity equal to the casting speed was imposed on the wide and narrow faces to represent the motion of the shell drag in a fixed frame of reference, and 15 s of instantaneous results were collected. After that, the buoyancy force was switched on and a further 15 s of simulation was carried out. For the different sets of conditions, we cannot identify any significant change in the flow pattern or in the temperature distribution. So the drag of the shell and the buoyancy force do not seem to have much influence on the solution of the simulation. However, the drag velocity at the shell and the buoyancy force were kept during the remaining simulation to make it more close to a real case. Snapshots of the flow and temperature fields were collected at several cross sections of the domain every 100 time steps, i.e. 0.05 s. The flow and temperature fields were then animated to study the transient dynamics. The collection of statistics was initiated after the flow and temperature fields reached a fairly stationary state. Mean statistics were collected for 40 s, and during the last 20 s, RMS values were calculated while mean values were continuously updated. To study the effect of the sub-grid scale model, separate simulations were carried out with and without the SGS model activated. Mean and RMS statistics were collected in the same manner for both the simulations. The results of simulations with and without the SGS model share a number of common features. The following sections first describe the results of the simulation with the SGS model. The effects of the SGS model are discussed separately in a later section.

4.4.1 Instantaneous velocity and temperature fields

Transient features of the steel flow and heat transfer phenomena are very important to the understanding of the defect formation in continuous casting processes. By analyzing the snapshots and animations of the instantaneous flow and temperature fields, we can identify the structures and dynamics of the flow and temperature fields.

One instantaneous flow field in the center plane is plotted in Fig. 4.6. The jet coming out from the side port impinges on the narrow face and forms two large recirculation regions, the "upper roll" and the "lower roll". Small vortices can be seen at the edge of the jets due to shear with the surrounding steel. The upper roll region sometimes is comprised of one single large vortex, while at other times it breaks into several small vortices, which is also observed in previous work.^[2] A typical instantaneous temperature field in the center plane is shown in Fig. 4.7. The high temperature core diffuses very quickly. Around 150 mm out of the nozzle, the superheat carried by the jet drops from 57 °C to 30 °C. The upper roll recirculation keeps the region above the side jet at a fairly constant temperature (~ 20 °C above the liquidus temperature). Figure 4.8 shows typical instantaneous flow and temperature fields in the symmetry plane between narrow faces. The central jet comes out of the center port, spreads out quickly, touching the wide faces after going down for around 0.1 m.

The side jet impinges on the narrow face and splits into two wall jets, one going down to the outlet of the domain and the other going up to the top surface. Figure 4.9 and Fig. 4.10 show the instantaneous velocity and temperature fields in the cross section cutting through the wall jets, 17 mm and 93 mm from narrow face, respectively. Instantaneously, the wall jets show wide face to wide face oscillations. In the wall jet, the temperature is fairly uniform at around 25° C above the liquidus temperature. There are spots of cold fluid near the wide faces that are entrained by the small vortices near the shell. Also there is a small recirculation region at the corner of the top surface and narrow face, shown in Fig. 4.9 as the meniscus region near the top surface with small velocities and lower temperatures: only $5 \sim 10^{\circ}$ C above the liquidus temperature.

The jet has strong swirl, as shown in Fig. 4.11 (192 mm to NF). Even though that the geometry, mesh and initial conditions are perfectly symmetrical, the jet develops significant instantaneous asymmetry due to its turbulent nature. At times the vortices generated are symmetrical on both sides of the jet, but most of the times the jet has asymmetrical multiple swirls in it. The jets also oscillate from wide face to wide face. From an animation of the flow field, we can estimate the oscillation frequency to be on the order of $5 \sim 10 Hz$. The temperature field above the jet, which is in the upper roll region, is fairly uniform. The temperature below the jet is much lower due the cooling of the wide faces. Figure 4.12 shows a snapshot in the cross section closer to the center, 291 mm from the narrow face. This cross section cuts through the core of the jet, which has a very high temperature (57 °C superheat). The region below the jet in this plane is one of the coldest regions in the domain. The cooling from the wide faces and the lack of influence from the hot jets make this region only a few degrees above the liquidus temperature.

Snapshots of the velocity and temperature fields near the meniscus are shown in Fig. 4.13 and Fig. 4.14, 38.5 mm and 82 mm down from the top surface, respectively. Near the meniscus, there is strong flow going towards the SEN. When this flow approaches the SEN, it generates vortices around the nozzle. Due to the oscillations of the jet, the flow near the top surface has a wavy pattern that makes it oscillate between the wide faces. The temperature near the top surface is fairly uniform, except at the cold spot caused by the entraining of vortices near the solidifying shell. Being closer to the jet, the flow field in the cross section 82 mm from the meniscus is affected more by the transient features of the jet. Alternative bursts of flow near the wide faces caused by the oscillation of the jet are observed. There are also more vortices near the narrow face compared to the cross section closer to the top surface.

One snapshot cutting through the side jet (161 mm below the meniscus) is shown in Fig. 4.15. The jet generates vortices as soon as it exits the side port. There are strong vortices near the narrow face where the wall jet goes up, indicating the helical motion in the wall jet. The recirculation flow generates vortices in the vicinity of the SEN, which is also seen in the cross sections closer the top surface. Another snapshot which cuts through both the central jet and the side jet is shown in Fig. 4.16, 243 mm from the top surface. Several small eddies are observed in the central jet. The side jet exhibits strong wide face to wide face oscillations, as discussed before, with the oscillation frequency being on the order of $5 \sim 10Hz$. There are strong vortices near the narrow face, caused by the confinement of the jet impingement. The length scale of these vortices are generally short lived, lasting around 0.5 s. The temperature in the core of the jet is very high (57 °C superheat) and there is a cold region between the side jet and the central jet. The temperature in the recirculation region is fairly uniform (20 °C superheat).

The instantaneous flow and temperature fields near the impingement point are shown in Fig. 4.17 and Fig. 4.18 (339 mm and 445 mm from the meniscus). The central jet induces several secondary vortices, which help rapid diffusion of the central jet. Near the narrow face, there are strong vortices generated by the impingement of the jet. The vortices vary in length scale and life span, with the largest vortex around 0.1 m in diameter and the duration of the vortices less than 1 s. The temperature in the wall jets along the narrow face is roughly constant. The temperature between the wall jet and the center jet is much lower, as it is cooled on both sides by the wide faces and has no entrainment of the hot jets. The vortices in the wall jet, side jet and central jet enhance mixing, making the temperature in these regions fairly constant. The vortices in the wall jet, combined with the large recirculating flow make the temperature in the upper roll region stay almost constant.

Further down in the domain, the flow is dominated by the vortices in the wall jet and central jet. These vortices are the main driving mechanism for the mixing in the lower region of the mold. Snapshots at 741 mm and 944.5 mm from the top surface are shown in Fig. 4.19 and Fig. 4.20. In the animation of the flow fields, we observed large vortices to break down into small vortices and small vortices later merging into large ones. The vortices in these regions last longer than the ones in the top region of the mold, because they are further away from the effects of the fast transient features of the jets. The temperatures in the center jet region and in the wall jet regions are fairly uniform at 25°C higher than the liquidus temperature. The temperature difference between the center jet and the wall jet is greatly influenced by the transport of the vortices. When a strong vortex moves into the region, it brings steel with higher temperature with it and makes the temperature in the region rise. Otherwise, the region remains at a low temperature due to the cooling from the wide faces.

4.4.2 Mean velocity and temperature fields

The flow and temperature fields were averaged over time after reaching a statistically stationary state. The solution for each case with or without the SGS model was averaged for 40 s of simulation time. Though fluctuations over larger time periods may exist, 40 s of averaging can give sufficiently accurate mean flow and temperature fields.

The mean velocity field and a streak line plot in the center plane are shown in Fig. 4.21 and Fig. 4.22. The jet coming out of the side port forms a typical two roll flow pattern. The swirls in the nozzle (Fig. 4.70) affected the shape of the jet in the center plane making it wider near the nozzle outlet. The flow out of the central port acts much like a free jet except that it is bounded by the two wide faces. The central jet and the recirculating flow in the lower roll form another recirculation region. The streak line plot demonstrates the flow pattern more clearly. In Fig. 4.22, we can see clearly the position and extent of the recirculation region as well as the jet angle and impingement point position. The geometrical impingement point, defined as the intersection of the jet axis and the NF, is at z = 1.02 m (46 mm below meniscus). The stagnation point, where the jet bifurcates, is at z = 0.96 m (40 mm below the)top surface). The downward angle between the jet and the horizontal line is 34° . The position of the eye of the upper roll is 0.3 m down from the top surface and 0.36 mleft from the narrow face. The eye of the lower roll is out of the simulation domain. One interesting feature of the flow field is that there is a very small vortex at the top right corner of the domain. This vortex is caused by the turning of the wall jet going upwards and the downward movement of the dragging shell. It can be inferred that the larger the casting speed, the larger this recirculation region will be. The presence of this vortex greatly affects the heat transfer in the corner region.

The time-average flow fields in cross sections parallel to the narrow face are shown in Fig. 4.23 to Fig. 4.28. The collision of the jet and the recirculating flow forms regions with swirl. Two symmetrical swirls can be seen Fig. 4.24. After the jet expands and takes up the whole thickness of the domain, the recirculation flow can not pass around the jet any more, so two swirls below the jet are formed. The swirls on the side of the jet still exist, as shown in Fig. 4.25. The side jet bifurcates near the narrow face, and splits into two wall jets which can be seen in Fig. 4.26 to Fig. 4.28. A small recirculation region near the meniscus can be identified in Fig. 4.28.

Mean flow fields in the cross sections parallel to the top surface are shown in Fig. 4.29 to Fig. 4.36, ordered with increasing distance from the meniscus. The mean flow towards the SEN near the top surface is strong and fairly uniform, as shown in Fig. 4.29 and Fig. 4.30. When the flow goes past the SEN, there is flow separation at the corner of the SEN and vortices are generated. The jet and the recirculation flow in the upper roll forms vortices on both sides of the jet, which can be seen in Fig. 4.31. Due to the confinement by the wide faces, the flow can not expand freely after impinging on the narrow face. The flow is forced to turn direction at the WF-NF corner and forms a spiral structure. The helical flow shows up as vortices at the WF-NF corner in the cross section parallel to the meniscus. These vortices can be seen in many of the cross sections. The center jet does not have significant flow in the plane perpendicular to its direction. But it induces secondary vortices around it, which will become one of the mechanisms that enhances mixing further down in the domain (Fig. 4.35). 3-D stream traces shown in Fig. 4.37 illustrates the overall picture of the average flow field. Three large recirculation regions are clearly shown: the upper recirculation zone and two counter rotating lower recirculation zones found between the central and side jets. The twisting of the stream trace ribbon indicates spiral motion. There are symmetrical swirls above the jet coming out of the side port. The helical structures in the flow at the WF-NF corner are mainly due to the confinement of the impinging jet.

The time-average temperature field in the center plane is shown in Fig. 4.38. The jets have hot cores at the casting temperature, which diffuse quickly. Most of the jet is roughly at a temperature 30°C above the liquidus temperature of steel. The temperature in the upper roll region is quite uniform and has a superheat of 20°C. In the small recirculation region at the top right corner, fluid is much colder, only 5°C over the liquidus temperature. The temperature distribution in the whole domain can be seen in Fig. 4.39 and Fig. 4.40. The temperature fields shown in these cross sections confirm the observation that there are large regions in the wall jet, central jet and side jet where the temperature is fairly uniform around 25°C above the liquidus temperature. The coldest regions in the temperature field are places very close to the solidifying shell and the region between the wall jet and the center jet, low in the domain. In these regions the steel is cooled from both wide faces and there is no strong flow to mix it with higher temperature fluid. Also the cold region at the corner of the top surface and the narrow face caused by the small recirculation region at the corner is detrimental to the process and may cause serious problems when excessively cooled. This may be important for the formation of "hooks" or meniscus solidification to surface defects.^[14]

4.4.3 RMS statistics of velocity and temperature field

RMS statistics were collected after 20 s of mean values have been computed. While updating the mean value, the RMS statistics over a 20 s time period were calculated. This method for collecting RMS was used for both the cases with and without SGS model.
Figure 4.41, Fig. 4.44 and Fig. 4.47 show the RMS statistics for u, v and w velocity components in the center plane. The largest velocity fluctuation appears at the edge of the jet. The maximum u RMS appears above the side jet while the maximum wRMS is below the jet. Velocity fluctuation in the jet region is also very large. The RMS of u and w velocity decreases near the impingement region, but v RMS remains quite large in the impingement region, indicating strong WF-WF oscillations. A comprehensive picture of the velocity RMS statistics in the 3-D domain are shown in Fig. 4.42, Fig. 4.43, Fig. 4.45, Fig. 4.46, Fig. 4.48 and Fig. 4.49. The highest velocity fluctuation is at the edge of the jet, where strong vortices are generated. There are two regions in the nozzle that have a large velocity fluctuation. On top of the central port, where the fluid goes through a contraction into the port, the fluctuation of the v and w velocity components is very large. At the top of the side port, where the flow turns direction at a sharp corner, the u RMS is very large.

Like the RMS statistics of velocity, the RMS of temperature is largest at the edges of the jets. Figure 4.50 shows the RMS of temperature in the center plane. Figure 4.51 and Fig. 4.52 show the distributions of temperature RMS in the whole domain. The RMS of temperature peaks at the edges of the jet at a value of 14 °C. In the upper roll and wall jet regions, the fluctuations of temperature are generally below 3 °C. Near the solidifying shell, the temperature fluctuation is around 5 °C. The fluctuation of temperature by several degrees will greatly change the heat flux to the solidifying shell. The turbulence fluctuations play an important role in the heat transfer rate to the solidifying shell.

4.4.4 Comparison of velocity profile with dye injection experiment

A full-scale water model of the thin-slab caster studied in this work was constructed^[87] to study flow in the liquid pool. The flow field was visualized by injecting dye into the submerged entry nozzle. Flow velocities were estimated through analysis of successive frames of videotape used to track movement of the die front, knowing the times between the frames.^[88] The videotaped flow patterns could then be compared qualitatively and quantitatively with the simulation results.

The mean flow patterns with and without the SGS model are compared with the experiment in Fig. 4.54 and Fig. 4.53. In these figures, the mean velocity fields from the numerical simulation are overlapped onto one of the dye injection experiment snapshots. The flow pattern in the simulation matches the experiment qualitatively, with a similar jet angle and width.

The profile of velocities is also compared against the velocity measurements by tracking the positions of the dye front.^[88] This experimental method has some intrinsic uncertainties. The dye front is some sort of average through the whole thickness of the water model. And also the method ignores the diffusion of the dye. However, the data obtained using this method can still shed some light on the credibility of the numerical simulation. The velocity profile along the jet is compared in Fig. 4.55 (error bar is RMS of velocity). The simulation results match the measurements very well. Velocities towards the SEN near the top surface are compared in Fig. 4.56. The simulation results under-predict the velocity in this region. This shows that the top surface velocity is very sensitive to minor changed in the jet impingement features.

From these limited comparisons with experiments, we can conclude that the computed flow field is reasonable.

4.4.5 Comparison of temperature profile with plant measurements

The temperature profiles of the simulation in the upper roll region are compared against plant measurements^[86,87] in the same caster with the same casting condition as the simulation. The schematic of the apparatus is shown in Fig. 4.1. Table 4.1 gives the details of the five sets of measurements. The comparisons between simulation results and experimental data are shown in Fig. 4.57 to Fig. 4.61. In these figures, the error bars are the RMS values of temperature which show the range of temperature fluctuation. The temperature profile in the simulations match the experiment very well with the exception of the profile 125 mm from the narrow face. Considering the fact that it is not very likely that the flow going from narrow face towards the SEN could have higher temperature at both the narrow face side and the SEN side and have lower temperature in between, it is most probably that inaccuracy in the experimental data caused this discrepancy. From the other four sets of data, we can see that the temperature in the upper roll region remains roughly constant around $1520 \,^{\circ}C$ (18 $^{\circ}C$ of superheat). The temperature fluctuation is around $\pm 4 \,^{\circ}C$. The match between simulation results and experimental data indicates that the temperature solution is accurate.

4.4.6 Heat transfer rate to the solidifying shell

The instantaneous heat flux to the solidifying shells is shown in Fig. 4.62. The instantaneous heat flux to the narrow face reaches over $1,800 \, kW/m^2$. The peak heat flux to the wide faces reaches over $900 \, kW/m^2$. The positions of peak instantaneous heat flux oscillates as the jet wobbles and changes the impingement point. The peak position will oscillate from wide face to wide face as well as in the casting direction. Multiple heat flux peaks to the narrow face are observed at the same instance. This is caused by the small turbulent eddies in the impingement region. The heat flux to the wide faces is the largest along the jets and near the narrow face. The side jet and the central jet expand as they travel, and cause heat flux peaks when they touch the wide faces. The vortices generated by the jet impingement at the corner formed by wide faces and narrow face contribute to the high heat flux in the wide faces near the narrow face.

Figure 4.63 shows the average heat flux to the shells. The maximum mean heat flux is $750 \, kW/m^2$ to the narrow face and $450 \, kW/m^2$ to the wide faces. The high heat flux region in the narrow face is not positioned symmetrically around the impingement point. The high heat flux region extends much more to the lower side of the impingement point, which was also observed in the experimental study of oblique impinging jet by Goldstein.^[40] In the wide faces, there are large areas with average heat flux over $400 \, kW/m^2$. As discussed above, the high heat flux is caused by the jets touching the wide faces and the vortices at the WF-NF corner. In the narrow face, the heat flux decreases away from the impingement point. But the heat flux rises to around $200 \, kW/m^2$ near the top surface, which is the due to the small recirculation region at the corner of top surface and narrow face. The upper roll recirculation flow gives relatively higher heat flux to the wide faces near the meniscus ($\sim 200 \, kW/m^2$). Also, the vortices generated as the flow goes around the SEN raise the heat flux to the wide faces to around $200 \, kW/m^2$ near the nozzle. The RMS statistics of heat flux are plotted in Fig. 4.64. The heat flux fluctuation can be as high as half of its time-average value. The shape of the RMS contours are similar to that of the mean heat flux.

Different profiles of heat flux in the narrow face and wide faces are plotted to give a more clear illustration. Mean and RMS heat flux along the centerline of the narrow face are shown in Fig. 4.65. The mean heat flux for cases with and without the SGS model differ in peak value by $100 \, kW/m^2$ and in position by $50 \, mm$. The RMS heat flux is very close for both cases. RMS heat flux reaches as higher as $350 \, kW/m^2$, nearly 50% of the maximum mean value. Figure 4.66 shows the mean and RMS heat flux along the line in wide faces $20 \, mm$ from the narrow face. The maximum is $450 \, kW/m^2$ for mean heat flux and $100 \, kW/m^2$ for RMS heat flux. The RMS heat flux does not have large variations, and has a range of 50 to $100 \, kW/m^2$. The mean and RMS heat flux in the wide faces along the side jet and the central jet are shown in Fig. 4.67 and Fig. 4.68. The peak mean heat transfer rate is $450 \, kW/m^2$. The highest RMS heat transfer rate is $150 \, kW/m^2$. Near the wide faces, the SGS model has no significant effect on the heat transfer solution.

4.4.7 Effect of sub-grid scale model on the solution

The inlet condition is very important to the flow and temperature field solution. The interest of this work is on the flow and heat transfer in the mold region. The attached nozzle domain is an effort to give the inlet condition into the mold region as accurate as possible. Fig. 4.69 shows the averaged flow field of the nozzle side port for the simulation without the SGS model. It is shown that the flow is mainly restricted to the center of the port. The velocities are very small at the bottom of the port. Near the top of the side port, there is recirculating flow from the mold into the nozzle. On both sides of the nozzle, there are swirl between the jet and the nozzle wall. Fig. 4.70 shows the same velocity field plot for the standard case with the SGS model. The flow field shares the same feature as that in the case without the SGS model. The difference is that the swirl is stronger for the case with the SGS model. This can be explained by the additional sub-grid scale viscosity. The swirl region extends all the way up to the top of the side port. Other than this, the two flow fields are quite similar. The mass flow through the side ports is almost the same for both cases. The flow through the side port comprises 84.58% and 84.85% of the total flow rate for cases with and without the SGS model, respectively.

The instantaneous flow and temperature fields are very similar between the cases with and without the SGS model. This can be shown by comparing Fig. 4.6 and Fig. 4.7 with Fig. 4.71 and Fig. 4.72. The only visible difference is that the jet in the center plane is wider right out of the nozzle for the case with the SGS model. From the discussion above we can see clearly that the wider jet in the center plane is caused by the stronger swirls at the nozzle outlet. This difference also shows up in the mean velocity field, as shown by comparing Fig. 4.21 and Fig. 4.73. Aside from these minor differences, the mean velocity fields are almost identical between the two cases. This can be seen more clearly in the streak line plot (Fig. 4.22 and Fig. 4.74). The angle of the jet, the position of the eye of the upper roll and the impingement point are the same for both cases. The mean temperature solutions for the two cases also have little difference. The temperature fields shown in Fig. 4.75 to Fig. 4.77 for the case without the SGS model are almost the same as those for the case with the SGS model, as shown in Fig. 4.38 to Fig. 4.40, aside from the swirl effect originated from the nozzle which showed up in the temperature field in the center plane. Velocity and temperature profiles at different x positions in the center plane are compared for cases with and without the SGS model in Fig. 4.78 to Fig. 4.81. The difference between the two case are small. This again proves the conclusion that the SGS model does not affect the mean solution very much. Comparisons of mean velocity profiles are smalls in Fig. 4.55 to Fig. 4.61 all show that the SGS model results are not much different from the solution without the SGS model.

The RMS statistics between the cases with and without the SGS model are also comparable in magnitude as well as their distributions. The velocity RMS statistics for the simulation without the SGS model are shown in Fig. 4.82 to Fig. 4.90. There is not much difference between their counterparts in Fig. 4.41 to Fig. 4.49. The part which differs most is the region above the jet, near the top of the side port exit. The difference again originates from the nozzle outlet. Figure 4.94 and Fig. 4.95 show the velocity fluctuations very close to the narrow face. The RMS of u velocity components, which is normal to the narrow face, is almost ten times smaller than the RMS of the other two velocity components. From the figures we can see that the v and w RMS are both larger in the case with the SGS model, which indicates stronger turbulence. This is believed to be the reason for the higher heat flux to the narrow face in the case with the SGS model. As for RMS statistics of temperature, the solution for the case without the SGS model can be examined in Fig. 4.91 to Fig. 4.93. There is not much difference compared to the solution with the SGS model (Fig. 4.50 to Fig. 4.52).

The instantaneous, mean and RMS of heat flux to the solidifying shells in the simulation without the SGS model are shown in Fig. 4.96 to Fig. 4.98, and the results with the SGS model are shown in Fig. 4.62 to Fig. 4.64. Other than the mean heat flux to the narrow face, the two cases are very similar. The peak mean heat flux reaches $660 \, kW/m^2$ without the SGS model and $750 \, kW/m^2$ with the SGS model. The comparison of heat flux profiles on the solidifying shells can also be seen in Fig. 4.65 to Fig. 4.68, which show the similarity between the two cases.

There are some differences between the solutions of the cases with and without the SGS model. The mean flow and temperature near the nozzle side exit, the RMS statistics close to the narrow face and the mean heat flux to the narrow face all differ slightly. The difference in the solutions in the mold with and without the SGS model arises from the differences in the flow field and the turbulence characteristics in the nozzle region, as shown in Fig. 4.69 and Fig. 4.70. However, the solutions for the two cases are very close on the whole, which can be seen in the many comparisons of the two cases. Thus the SGS model does not appear to have a large influence on the solution.

4.4.8 Superheat budget

According to the boundary conditions of the simulation, superheat carried by the molten steel enters the domain through the nozzle inlet. The superheat in the domain has only two ways of leaving the domain: either through the solidifying shells or through advection across the bottom outlet of the domain. The superheat removal through the solidifying shells is of particular interest to this study. During the statistical stationary state of the simulation, the superheat entering and leaving the domain has a dynamic balance, which means the superheat entering the domain roughly equal to the superheat leaving the domain. Due to the transient nature of the flow and heat transfer, this is not necessarily true for one particular instant, but the energy budget must be balanced on average.

The solidification of steel is not modelled in this work, so the latent heat evolution of the liquid steel occurs outside the domain and can be neglected. However, the sensible heat of the liquid steel, i.e. superheat, is of interest metallurgically, so the total system energy was tracked. This superheat (E) is defined as following:

$$E = \int \rho C_P (T - T_0) dV \tag{4.1}$$

where T_0 is the liquidus solidification temperature of the steel.

The superheat coming into the domain is not necessarily balanced by the superheat going out of the domain instantaneously, which causes fluctuation of superheat in the whole system. Fig. 4.99 shows the total sensible heat variation with respect to time. The system superheat fluctuates about the mean value. The mean of total system superheat is $5.41 \times 10^6 J$ and $5.06 \times 10^6 J$ for the simulations without the SGS model and with the SGS model respectively. The difference is most probably caused by the different sampling times of the two cases. It is clear that the total system energy has frequencies of fluctuation lower than 0.025Hz and it is not surprising to have minor differences between the two averages over a 40 s time period. The magnitude of the fluctuation is around $\pm 4 \times 10^5 J$, which is 8% of the average value.

The detailed superheat budget can be calculated using the mean flow and temperature field. The superheat enters or leaves the domain through the inlet and outlet only by advection according to the boundary conditions and can be calculated by:

$$W = \int \rho C_P (T - T_0) V dA \tag{4.2}$$

where V is the velocity normal to the inlet or outlet surface. Notice that in the above equation, the average superheat cannot be simply calculated using the multiplication of average temperature and average velocity. If we derive the equation using the Reynolds-average method, i.e. decompose the variables into average and fluctuation components, an extra term of $\overline{T'V'}$ appears. This term represents the turbulent heat flux entering or leaving the domain. Because there is no temperature variation at the inlet, this term will be zero at the inlet. But it is a significant part of the superheat leaving the domain at the outlet.

The energy leaving through the solidifying shell can be calculated by integrating the heat flux over the narrow face or wide faces.

$$W = \int q dA \tag{4.3}$$

The total superheat coming into the domain is $449 \, kW$. For the case of simulation without the SGS model, the superheat leaving from the narrow face is $44 \, kW$. The superheat leaving from the two wide faces is $111 \, kW$ and $106 \, kW$, respectively. The superheat leaving the domain through the outlet is $160 \, kW$ for convection and $29 \, kW$ for turbulent heat flux. Although the heat flux at the narrow face has a larger peak than that at the wide faces, because of the larger area of the wide faces, the superheat leaving through the wide faces is much larger than that through the narrow face. The surface area ratio of the narrow face to wide face is 1 to 7.45, the ratio of superheat removed is 1 to 4.93, and so the narrow face has a larger average heat flux. For the case of simulation with the SGS model, the superheat leaving through the narrow face is $55 \, kW$. The superheat leaving through the domain outlet is $126 \, kW$ by convection and $35 \, kW$ by turbulent heat flux fluctuation. As shown in Fig. 4.63, Fig. 4.97 and Fig. 4.65, the average heat fluxes to the narrow face in the case with the SGS model is higher

than that in the case without the SGS model, with a peak heat flux 25% higher. This caused the increase of superheat going through the narrow face in the case with SGS model. On the other hand, there is not much difference in the average heat flux to the wide faces (Fig. 4.63, Fig. 4.97 and Fig. 4.66 to Fig. 4.68). The superheat leaving through the wide faces remains approximately the same for the two cases. As a result, the superheat leaving through the outlet is less in the case with the SGS model.

In the real casting process, there is also heat loss through the liquid flux layer on top of the molten steel. With the work in Chapter 2 and the fluid flow solution in this work, it is possible to estimate the heat loss through the top surface. It is known that the liquid flux layer in this caster is $5 \sim 10 \, mm$ (Table 4.1). Using the flux properties given in Table 2.4, we can calculate that the heat flux for pure conduction is $330 \, kW/m^2$ assuming the liquid flux layer is $5 \, mm$ thick. For the flux layer thickness range, there is no natural convection in the layer. Given the average steel velocity at top surface around 0.1 m/s, it can be inferred from Fig 2.33 that the Nu will be 1.1. Multiply the Nu, heat flux for conduction and the area of top surface $(0.0563 m^2)$, we then get the super heat leaving the top surface to be about $20 \, kW$, i.e. 4.5% of the total superheat coming into the domain. There are some uncertainties in this calculation. A thinner flux layer will have larger velocity gradients and thus larger shear stress at the steel/flux interface, which then result in a lower bottom shear velocity and smaller Nu. A smaller width of the flux layer will make the end effect of forced convection more prominent, resulting in a larger Nu. Taking into consideration the uncertainties, we can estimate that the superheat leaving the domain to have a range of $2\% \sim 6\%$ of the total superheat.

Figure 4.100 summarizes the superheat budget in this caster. About 64% of the superheat leaves through the solidifying shells. Around 35% of the superheat leaves through the domain outlet through advection. In a real caster, the shell will grow as it is dragged downwards. So the actual shell surface area will be larger than the case

of a straight plane assumed here. Assuming that the heat flux will not be affected significantly by the shell growth, then the superheat leaving though the solidifying shells will be larger in a real operation.

4.5 Summary

Large eddy simulation of the flow and heat transfer in the liquid steel pool region has been carried out. The results of the simulation agree well with the dye injection experiments in a full-scale water model and the plant measurements of temperature in the upper roll region. The SGS model has only a minor impact on the flow and heat transport characteristics of the simulation.

Animations of the flow and temperature field give knowledge of the transient flow structures and heat transport. The side jet shows strong wide face to wide face oscillations with frequencies on the order of $5 \sim 10 Hz$. Although the domain and grids are perfectly symmetrical, asymmetric jet flow was observed. Vortices were observed at the WF-NF corner, but the life span of the spirals are generally short, less that 1 s. On average, the wall jets also flow in a spiral motion due to confinement of the wide faces. The swirls in the wall jets help mixing and keep the temperature fairly constant. The upper roll recirculation also helps to create a fairly uniform temperature field above the jet.

A small recirculation region is observed in the simulation at the corner of the narrow face and the top surface. The tuning of the wall jet combined with the downward dragging of the shell generates this recirculation region. This recirculation region is a region with low temperature. The recirculating flow impinges onto the narrow face and locally raises the heat transfer rate to the shell. The region is significant to the process in that it is where defects tend to form.

The highest instantaneous heat transfer rate reaches $1,800 \, kW/m^2$ in the impingement region of the side jet. The average heat flux peak to the narrow face is $750 \, kW/m^2$. The jets also create regions of locally high heat transfer rate on the wide faces with peak mean value of $450 \, kW/m^2$. The RMS fluctuation of heat flux to the narrow face reaches as high as $350 \, kW/m^2$. Twelve of the superheat is extracted from the narrow face of this $132 \, mm$ -thick caster. Sixty four of the superheat is removed in the mold.

4.6 Figures and Tables



Fig. 4.1: Equipment used in measuring temperature in the liquid pool



Fig. 4.2: Example of temperature measurement data



Fig. 4.3: Simulation domain and and boundary conditions



Fig. 4.4: Grids in the nozzle



Fig. 4.5: Grids in the mold



Fig. 4.6: Instantaneous velocity vector plot of center plane (with SGS model)



Fig. 4.7: Instantaneous temperature field of center plane (with SGS model)



Fig. 4.8: Instantaneous velocity and temperature field of center plane between narrow faces (with SGS model)



Fig. 4.9: Instantaneous velocity and temperature field of cross section 17 mm from NF (with SGS model)



Fig. 4.10: Instantaneous velocity and temperature field of cross section 93 mm to NF (with SGS model)



Fig. 4.11: Instantaneous velocity and temperature field of cross section 192 mm to NF (with SGS model)



Fig. 4.12: Instantaneous velocity and temperature field of cross section 291 mm to NF (with SGS model)



Fig. 4.13: Instantaneous velocity and temperature field of cross section 38.5 mm from the top surface (with SGS model)



Fig. 4.14: Instantaneous velocity and temperature field of cross section 82 mm to the top surface (with SGS model)



Fig. 4.15: Instantaneous velocity and temperature field of cross section 161 mm to the top surface (with SGS model)



Fig. 4.16: Instantaneous velocity and temperature field of cross section 243 mm to the top surface (with SGS model)



Fig. 4.17: Instantaneous velocity and temperature field of cross section $339 \ mm$ to the top surface (with SGS model)



Fig. 4.18: Instantaneous velocity and temperature field of cross section 445 mm to the top surface (with SGS model)



Fig. 4.19: Instantaneous velocity and temperature field of cross section 741 mm to the top surface (with SGS model)



Fig. 4.20: Instantaneous velocity and temperature field of cross section 944.5 mm to the top surface (with SGS model)



Fig. 4.21: Time averaged velocity field (with SGS model)



Fig. 4.22: Streamlines of mean velocity field (with SGS model)



Fig. 4.24: Mean velocity field of cross section parallel to the narrow face (291 mm from narrow face, with SGS model)



Fig. 4.25: Mean velocity field of cross section parallel to the narrow face (192 mm from narrow face, with SGS model)



Fig. 4.26: Mean velocity field of cross section parallel to the narrow face (93 mm from narrow face, with SGS model)



Fig. 4.27: Mean velocity field of cross section parallel to the narrow face (17 mm from narrow face, with SGS model)



Fig. 4.28: Mean velocity field of cross section parallel to the narrow face (0.35 mm from narrow face, with SGS model)



Fig. 4.29: Mean velocity field of cross section parallel to the top surface (38.5 mm from top surface, with SGS model)



Fig. 4.30: Mean velocity field of cross section parallel to the top surface (82 mm from top surface, with SGS model)



Fig. 4.31: Mean velocity field of cross section parallel to the top surface (161 mm from top surface, with SGS model)



Fig. 4.32: Mean velocity field of cross section parallel to the top surface (243 mm from top surface, with SGS model)



Fig. 4.33: Mean velocity field of cross section parallel to the top surface (339 mm from top surface, with SGS model)



Fig. 4.34: Mean velocity field of cross section parallel to the top surface (445 mm from top surface, with SGS model)



Fig. 4.35: Mean velocity field of cross section parallel to the top surface (741 mm from top surface, with SGS model)



Fig. 4.36: Mean velocity field of cross section parallel to the top surface (944.5 mm from top surface, with SGS model)



Fig. 4.37: 3-D stream traces of the time-averaged flow field



Fig. 4.38: Mean temperature field of center plane (with SGS model)


Fig. 4.39: Mean temperature fields in cross sections parallel to the narrow face (with SGS model) $\,$



Fig. 4.40: Mean temperature fields in cross sections parallel to the top surface (with SGS model) $\,$



Fig. 4.41: RMS u velocity in the center plane (with SGS model)



Fig. 4.42: RMS u velocity in the cross sections parallel to the narrow face (with SGS model)



Fig. 4.43: RMS u velocity in the cross sections parallel to the top surface (with SGS model)



Fig. 4.44: RMS v velocity in center plane (with SGS model)



Fig. 4.45: RMS v velocity in the cross sections parallel to the narrow face (with SGS model)



Fig. 4.46: RMS v velocity in the cross sections parallel to the top surface (with SGS model)



Fig. 4.47: RMS w velocity in center plane (with SGS model)



Fig. 4.48: RMS w velocity in the cross sections parallel to the narrow face (with SGS model)



Fig. 4.49: RMS w velocity in the cross sections parallel to the top surface (with SGS model)



Fig. 4.50: RMS temperature in the center plane (with SGS model)



Fig. 4.51: RMS temperature in the cross sections parallel to narrow face (with SGS model) $\,$



Fig. 4.52: RMS temperature in the cross sections parallel to the top surface (with SGS model)



Fig. 4.53: Mean flow field of simulation overlapped on the dye injection experiment (without SGS model)



Fig. 4.54: Mean flow field of simulation overlapped on the dye injection experiment (with SGS model)



Fig. 4.55: Velocity magnitude along jet axis compared with dye injection measurements



Fig. 4.56: Top surface velocity compared with dye injection measurements



Fig. 4.57: Comparison of simulation mean temperature profile and plant measurement (measurement 1)



Fig. 4.58: Comparison of simulation mean temperature profile and plant measurement (measurement 2)



Fig. 4.59: Comparison of simulation mean temperature profile and plant measurement (measurement 3)



Fig. 4.60: Comparison of simulation mean temperature profile and plant measurement (measurement 4)



Fig. 4.61: Comparison of simulation mean temperature profile and plant measurement (measurement 5)



Fig. 4.62: Instantaneous heat fluxes through three solidifying faces (with SGS model) $\ensuremath{\mathsf{model}}\xspace$



Fig. 4.63: Time averaged heat fluxes through three solidifying faces (with SGS model) $\,$



Fig. 4.64: Heat fluxes fluctuations through three solidifying faces (with SGS model)



Fig. 4.65: Heat flux along the center line of narrow face



Fig. 4.66: Heat flux along the line in wide face (20 mm from narrow face)



Fig. 4.67: Heat flux in the wide faces along the side jet



Fig. 4.68: Heat flux in the wide faces along the center jet



Fig. 4.69: Mean velocity field of the side nozzle port (without SGS model)



Fig. 4.70: Mean velocity field of the side nozzle port (with SGS model)



Fig. 4.71: Instantaneous velocity vector plot in the center plane (without SGS model)



Fig. 4.72: Instantaneous temperature field in the center plane (without SGS model)



Fig. 4.73: Time averaged velocity field (without SGS model)



Fig. 4.74: Streamlines of mean velocity field (without SGS model)



Fig. 4.75: Mean temperature field in the center plane (without SGS model)



Fig. 4.76: Mean temperature fields in cross sections parallel to the narrow face (without SGS model)



Fig. 4.77: Mean temperature fields in cross sections parallel to the top surface (without SGS model)



Fig. 4.78: Mean velocity and temperature profile in the center plane at x = 0.1 m



Fig. 4.79: Mean velocity and temperature profile in the center plane at x = 0.2 m


Fig. 4.80: Mean velocity and temperature profile in the center plane at x = 0.3 m



Fig. 4.81: Mean velocity and temperature profile in the center plane at x = 0.4 m



Fig. 4.82: RMS u velocity in the center plane (without SGS model)



Fig. 4.83: RMS u velocity in the cross sections parallel to the narrow face (without SGS model)



Fig. 4.84: RMS u velocity in the cross sections parallel to the top surface (without SGS model)



Fig. 4.85: RMS v velocity in the center plane (without SGS model)



Fig. 4.86: RMS v velocity in the cross sections parallel to the narrow face (without SGS model)



Fig. 4.87: RMS v velocity in the cross sections parallel to the top surface (without SGS model)



Fig. 4.88: RMS w velocity in the center plane (without SGS model)



Fig. 4.89: RMS w velocity in the cross sections parallel to the narrow face (without SGS model)



Fig. 4.90: RMS w velocity in the cross sections parallel to the top surface (without SGS model)



Fig. 4.91: RMS temperature in the center plane (without SGS model)



Fig. 4.92: RMS temperature in the cross sections parallel to narrow face (without SGS model)



Fig. 4.93: RMS temperature in the cross sections parallel to the top surface (without SGS model)



Fig. 4.94: RMS velocity in the cross section parallel to the narrow face (0.35 mm from narrow face, without SGS model)



Fig. 4.95: RMS velocity in the cross section parallel to the narrow face (0.35 mm from narrow face, with SGS model)



Fig. 4.96: Instantaneous heat fluxes through three solidifying faces (without SGS model) $\,$



Fig. 4.97: Time averaged heat fluxes through three solidifying faces (without SGS model)



Fig. 4.98: Heat fluxes fluctuations through three solidifying faces (without SGS model) $\,$



Fig. 4.99: System internal energy variation with time



Fig. 4.100: Superheat removal percentage distribution

No.	Position of Measurement	Casting Temperature	Thickness of Powder (mm)	Thickness of Flux (mm)
1	Midway of NF and SEN $(295 \ mm \ from \ CL)$	1832 K (2838 F)	60	6
2	50 mm from SEN (150 mm from CL)	1831 K (2836 F)	62	10
3	50 mm from NF (440 mm from CL)	1831 K (2836 F)	83	5
4	125 mm from SEN (225 mm from CL)	1831 K (2836 F)	68	7
5	125 mm from NF (365 mm from CL)	1824 K (2824 F)	68	5

Table 4.1: Plant experiments conditions

Bottom Nozzle port diameter (mm)	32
Domain length (mold part) (m)	1.2
Total domain length (m)	1.76
Domain thickness (mm)	132
Domain width (mm)	492
SEN submerge depth (mm)	127
Nozzle inlet diameter (mm)	70
Side Nozzle port height (mm)	75
Side Nozzle port width (mm)	32
Casting speed $(mm \cdot s^{-1})$	24.5
Casting temperature (K)	1832
Solidus temperature (K)	1775
Laminar viscosity $(kg \cdot m^{-1} \cdot s^{-1})$	0.00555
Thermal conductivity $(W \cdot m^{-1} \cdot \mathbf{K}^{-1})$	26
Liquid d steel density $(kg\!\cdot\!m^{-3})$	7020
Specific heat of liquid steel $(J \cdot kg^{-1} \cdot \mathbf{K}^{-1})$	680
Thermal expansion coefficient (K^{-1})	$1.0\!\times\!10^{-4}$
Gravity constant $(m \cdot s^{-2})$	9.8
Laminar prandtl number	0.1452
Turbulent prandtl number	0.9

Table 4.2: Parameters and material properties for the simulation

Chapter 5. Conclusions

5.1 Flow and Heat Transfer in Molten Flux Layer

Computational models are used to simulate 2-D fluid flow and heat transfer in the liquid flux layer above a molten metal surface, such as encountered in the continuous casting of steel. The model includes the effects of natural convection, temperature-dependent viscosity, and shear velocity across the bottom surface. It is found that the Ra number for realistic liquid slag layers varies near the critical Ra number for the onset of natural convection. For fluxes with temperature-dependent viscosity, the variation of Nu with Ra is analogous to correlations for fluids with constant viscosity evaluated at the mean temperature, but the critical Ra number is larger. The increase in Nu number with layer thickness is also quantified for realistic fluxes.

For thin layers of realistic fluxes, natural convection is suppressed, so Nu increases linearly with increase of bottom shear velocity. The increase is greater with decreasing average viscosity. The increase of heat transfer above pure conduction is only due to end effects, and hence depends on the dimensions of the layer. For the flat interface shape investigated here, this increase is only one to three fold. Larger increases observed in practice could be due to phenomena not included in these computations, such as level fluctuations and flux consumption.

5.2 Impinging Jet Heat Transfer

LES simulations are carried out to study the heat transfer of a circular impinging jet on a planar surface. The simulation results are compared with available experimental data in the literature.

The LES model produces satisfactory results for heat transfer of a circular impinging jet. The predicted mean and RMS velocity are in reasonable agreement with the experimental data in the literature. The model captures the high heat transfer rate at the impingement point with the maximum difference between predicted value and measurements being around 20%. Thus the numerical method and computer program are validated for application to the continuous casting of steel.

5.3 Flow and Heat Transfer in Liquid Pool of Continuous Casting of Steel

Large eddy simulations of the flow and heat transfer in the liquid steel pool region have been carried out. The results of the simulations agree well with the dye injection experiments in a full-scale water model and the plant measurement of temperature in the upper roll region. The SGS model is observed to have little impact on the flow and heat transport characteristics.

Animations of the flow and temperature field give knowledge of the transient flow structure and heat transport. The side jet shows strong wide face to wide face oscillations with frequencies on the order of $5 \sim 10 Hz$. Although the domain and grids are perfectly symmetrical, asymmetric jet flow was observed. Vortices were observed at the WF-NF corner, but the life span of the spirals are generally short, less that 1 s. On average, the wall jets also flow in a spiral motion due to confinement of the wide faces. The swirls in the wall jets help mixing and keep the temperature fairly constant. The upper roll recirculation also helps to create a fairly uniform temperature field above the jet.

A small recirculation region is observed in the simulation at the corner of the narrow face and the top surface. The tuning of the wall jet combined with the downward dragging of the shell generates this recirculation region. This recirculation region is a region with low temperature. The recirculating flow impinging onto the narrow face increases the heat transfer rate to the shell. The region is significant to the process as it may be the location where defects tend to form.

The highest instantaneous heat transfer rate reaches $1,800 \, kW/m^2$ in the impingement region of the side jet. The time-average heat flux peaks on the narrow face at $750 \, kW/m^2$. The jets also create locally high heat transfer regions on the wide faces with a peak mean value of $450 \, kW/m^2$. The RMS fluctuation of heat flux to the narrow face reaches as high as $350 \, kW/m^2$. Twelve percent of the superheat is extracted from the narrow face of this $132 \, mm$ -thick caster. Sixty four percent of the superheat is removed in the mold.

5.4 Future Work

The effects of temperature dependent viscosity and the bottom shear velocity on the heat transfer in the liquid flux layer are studied in the present work. There are some other phenomena which need further investigation. The turbulent level fluctuation of the liquid flux layer is very important and not thoroughly understood. The three dimensional effect in the liquid flux layer also needs further study. A ambitious work would be to couple the simulation of steel flow and the flux layer.

The numerical model for simulating the flow and heat transfer in the caster mold has been proven successful. But further improvements can be made. More accurate treatment of the complex geometry can be achieved using immersed boundary method or boundary fitted coordinates. Parallel computing can decrease the execution time or make larger simulations feasible.

For the simulation of turbulent heat transfer in the mold region, more complex models including the shell and both sides of the caster can be applied in the future. More simulations are needed to investigate the different operating conditions. Multiphase flow simulations would also be helpful in modelling the process closer to real conditions.

References

- McDavid, R. M. and B. G. Thomas. "Flow and thermal behavior of the top surface flux/powder layers in continuous casting molds." *Metallurgical and Materials Transactions B* 27B, 672–685 (1996).
- 2. Sivaramakrishnan, S. Large Eddy Simulation, Particle Image Velocimetry in the Study of Mold Transients in Continuous Casting of Steel and Heat Transfer through Molten Slag Layers. Master's thesis, University of Illinois at Urbana-Champaign (2000).
- Schluter, A., D. Lortz and F. Busse. "On the stability of steady finite amplitude convection." J. Fluid Mech. 23, 129–144 (1965).
- Rossby, H. T. "A study of Benard convection with and without rotation." J. Fluid Mech. 36, 309–335 (1969).
- Koschmieder, E. L. and S. G. Pallas. "Heat transfer through a shallow, horizontal convecting fluid layer." Int. J. Heat Mass Transfer 17, 991–1002 (1974).
- Kirchartz, K. R. and H. Oertel Jr. "Three-dimensional thermal cellular convection in rectangular boxes." J. Fluid Mech. 192, 249–186 (1988).
- Goldstein, R. J., H. D. Chiang and D. L. See. "High-Rayleigh-number convection in a horizontal enclosure." J. Fluid Mech. 213, 111–126 (1990).
- Booker, J. R. "Thermal convection with strongly temperature-dependent viscosity." J. Fluid Mech. 76, 741–754 (1976).
- Mohamad, A. A. and R. Viskanta. "Laminar flow and heat transfer in Rayleigh-Benard convection with shear." *Phys. Fluids A* 4(10), 2131–2140 (1992).
- Tannehill, J. C., D. A. Anderson and R. H. Pletcher. Computational Fluid Mechanics and Heat Transfer, 2nd edition (Taylor & Francis, Washington, D.C., 1997).
- Boussinesq, J. "Théorie analytique de la chaleur." Paris: Gathier-Villars 2 (1903).
- 12. "FLUENT V6.1." Fluent Inc., Lebanon, NH. http://www.fluent.com (2003).
- 13. Dantzig, J. A. and I. Charles L. Tucker. *Modeling in Materials Processing* (Cambridge University Press, 2001).

- Thomas, B. G. "Continuous casting operation: Fluid flow." In Making, Shaping and Treating of Steel: Continuous Casting, A. Cramb, ed., Vol. 5, chapter 14.1-14.41 (AISE Steel Foundation, Pittsburgh, PA, 2003).
- 15. Larson, D. "Criteria for selecting mold powders to optimize continuous cast steel quality." *Industrial Heating* **53**, 16–19 (1986).
- Branion, R. V. "Mold flux for continuous casting." In *Steelmaking Proceedings*, Vol. 69, pp. 95–106 (Iron and Steel Society, Inc., Washington, D.C., USA, 1986).
- Pinheiro, C. A., I. V. Samarasekera and J. K. Brimacombe. "Mold flux for continuous casting of steel." *Iron and Steel Maker* 21, 12–14 (1994).
- Xie, B., J. Wu and Y. Gan. "Study on amount and scheme of carbon mixed in CC mold fluxes." In *Steelmaking Conference Proceedings*, Vol. 74, pp. 647–651 (Iron and Steel Society, Inc., Washington, D.C., 1991).
- Riboud, P. V. and M. Larrecq. "Lubrication and heat transfer in a continuous casting mold." In *Steelmaking Proceedings*, Vol. 62, pp. 78–92 (TMS/AIME, Detroit, MI, 1979).
- Lanyi, M. D. and C. J. Rosa. "Viscosity of casting fluxes used during continuous casting of steel." *Metallurgical and Materials Transactions B* 12B, 287–298 (1981).
- Mikrovas, A. C., S. A. Argyropoulos and I. D. Sommerville. "Heat transfer characteristics of molten slags." *Ironmaking and Steelmaking* 18(3), 169–182 (1991).
- Susa, M., K. Nagata and K. C. Mills. "Absorption coefficients and refractive indices of synthetic glassy slags containing transition metal oxides." *Ironmaking* and Steelmaking 20(5), 372–378 (1993).
- Taylor, R. and K. C. Mills. "Physical properties of casting powders: Part3 thermal conductivities of casting powders." *Ironmaking and Steelmaking* 15(4), 187–194 (1988).
- Iacovides, H. and B. E. Launder. "Computational fluid dynamics applied to internal gas-turbine blade cooling: a review." *International Journal of Heat and Fluid Flow* 16(6), 454–470 (1995).

- Kercher, D. S., J.-B. Lee, O. Brand, M. G. Allen and A. Glezer. "Microjet cooling devices for thermal management of electronics." *IEEE Transactions on Components and Packaging Technologies* 26(2), 359–366 (2003).
- Leland, J. E., R. Ponnappan and K. S. Klasing. "Experimental investigation of an air microjet array impingement cooling device." *Journal of Thermophysics* and Heat Transfer 16(2), 187–192 (2002).
- Zhang, X., Z. Yao, P. Hao and H. Xu. "Study on particle removal efficiency of an impinging jet by an image-processing method." *Experiments in Fluids* 32(3), 376–380 (2002).
- Polat, S. "Heat and mass transfer in impingement drying." Drying Technology 11(6), 1147–1176 (1993).
- Alvi, F. S., C. Shih, R. Elavarasan, G. Garg and A. Krothapalli. "Control of supersonic impinging jet flows using supersonic microjets." *AIAA Journal* 41(7), 1347–1355 (2003).
- Harman, T. B., J. M. Cimbala and M. L. Billet. "Reduction in size and unsteadiness of vtol ground vortices by ground fences." *Journal of Aircraft* **31**(3), 579–584 (1994).
- Woodruff, D. W. and J. M. Redwing. "Chemical vapor deposition of fine-grained equiaxed tungsten films." Surface & Coatings Technology 49(1-3), 215–220 (1991).
- Ghassemieh, E., M. Acar and H. K. Versteeg. "Improvement of the efficiency of energy transfer in the hydro-entanglement process." *Composites Science and Technology* 61(12), 1681–1694 (2001).
- Jambunathan, K., E. Lai, M. A. Moss and B. L. Button. "A review of heat transfer data for single circular jet impingement." Int. J. Heat and Fluid Flow 13(2), 106–115 (1992).
- Viskanta, R. "Heat transfer to impinging isothermal gas and flame jets." Experimental Thermal and Fluid Science 6, 111–134 (1993).
- Huang, L. and M. S. El-Genk. "Heat transfer of an impinging jet on a flat surface." Int. J. Heat Mass Transfer 37(13), 1915–1923 (1994).

- Elison, B. and B. W. Webb. "Local heat transfer to impinging liquid jets in the initially laminar, transitional, and turbulent regimes." Int. J. Heat Mass Transfer 37(8), 1207–1216 (1994).
- 37. Baughn, J. W., A. Hechanova and X. Yan. "An experimental study of entrainment effects on the heat transfer from a flat surface to a heated circular impinging jet." *Journal of Heat Transfer* **113**, 1023–1025 (1991).
- Hollworth, B. R. and L. R. Gero. "Entrainment effects on impingement heat transfer: Part II - local heat transfer measurements." *Journal of Heat Transfer* 107, 910–915 (1985).
- Yan, X. and N. Saniei. "Heat transfer from an obliquely impinging circular air jet to a flat plate." Int. J. Heat and Fluid Flow 18, 591–599 (1997).
- Goldstein, R. J. and M. E. Franchett. "Heat transfer from a flat surface to an oblique impinging jet." *Journal of Heat Transfer* 110, 84–90 (1988).
- Ma, C. F., Q. Zheng, H. Sun, K. Wu and K. Horii. "Local convective heat transfer from a vertical flat surface to oblique submerged impinging jets of large Prandtl number liquid." *Experimental Thermal and Fluid Science* 17, 238–247 (1997).
- Huang, L. and M. S. El-Genk. "Heat transfer and flow visualization experiments of swirling, nulti-channel, and conventinal impinging jets." *Int. J. Heat Mass Transfer* 41(3), 583–600 (1998).
- Colucci, D. W. and R. Viskanta. "Effect of nozzle geometry on local convective heat transfer to a confined impinging air jet." *Experimental Thermal and Fluid Science* 13, 71–80 (1996).
- 44. Shoukri, M. and A. Calka. "On the heat transfer characteristics of constrained air jets impinging on a flat surface." Int. J. Heat Mass Transfer 30(1), 205–208 (1987).
- Garimella, S. V. and R. A. Rice. "Confined and submerged liquid jet impingement heat transfer." *Journal of Heat Transfer* **117**, 871–877 (1995).
- Fitzgerald, J. A. and S. V. Garimella. "Flow field effects on heat transfer in confined jet impingingment." *Journal of Heat Transfer* 119, 630–632 (1997).
- 47. Garlimella, S. V. and B. Nenaydykh. "Nozzle-geometry effects in liquid jet impingement heat transfer." Int. J. Heat Mass Transfer 39(14), 2915–2923 (1996).

- 48. Sakakibara, J., K. Hishida and M. Maeda. "Vortex structure and heat transfer in the stagnation region of an impinging plane jet (simultaneous measurements of velocity and temperature fields by digital particle image velocimetry and laser induced fluorescence)." Int. J. Heat Mass Transfer 40(13), 3163–3176 (1997).
- 49. Sakakibara, J., K. Hishida and W. R. C. Phillips. "On the vortical structure in a plane impinging jet." J. Fluid Mech. 434, 273–300 (2001).
- 50. Hwang, S. D., C. H. Lee and H. H. Cho. "Heat transfer and flow structure in axisymmetric impinging jet controlled by vortex pairing." *Int. J. Heat and Fluid Flow* **22**, 293–300 (2001).
- Baughn, J. W. and S. Shimizu. "Heat transfer measurment from a surface with uniform heat flux and an impinging jet." *Journal of Heat Transfer* 111, 1096– 1098 (1989).
- Cooper, D., D. C. Jackson, B. E. Launder and G. X. Liao. "Impinging jet studies for turbulence model assessment I. flow-field experiments." *Int. J. Heat Mass Transfer* 36(10), 2675–2684 (1993).
- Craft, T. J., L. J. W. Graham and B. E. Launder. "Impimging jet studies for turbulence model assessment II. an examination of the performance of four turbulence models." *Int. J. Heat Mass Transfer* 36(10), 2685–2697 (1993).
- Behnia, M., S. Parneix and P. A. Durbin. "Prediction of heat transfer in an axisymmetric turbulent jet impinging on a flat plate." *Int. J. Heat Mass Transfer* 41(12), 1845–1855 (1998).
- Dianat, M., M. Fairweather and W. P. Jones. "Reynolds stress closure applied to axisymmetric, impinging turbulent jets." *Theoret. Comput. Fluid Dynamics* 8, 435–447 (1996).
- 56. Park, T. S. and H. J. Sung. "Development of a near-wall turbulence model and application to jet impingement heat transfer." Int. J. Heat and Fluid Flow 22, 10–18 (2001).
- 57. Behnia, M., S. Parneix, Y. Shabany and P. A. Durbin. "Numerical study of turbulent heat transfer in confined and unconfined impinging jets." *Int. J. Heat* and Fluid Flow 20, 1–9 (1999).

- Dianat, M., M. Fairweather and W. P. Jones. "Predictions of axisymmetric and two-dimensional impinging turbulent jets." Int. J. Heat and Fluid Flow 17, 530– 583 (1996).
- Laschefski, H., T. Cziesla and N. K. Mitra. "Evolution of flow structure in impinging three-dimensinal axial and radial jets." *International Journal for Numerical Methods in Fluids* 25, 1083–1103 (1997).
- Chung, Y. M., K. H. Luo and N. D. Sandham. "Numerical study of momentum and heat transfer in unsteady impniging jets." *Int. J. Heat and Fluid Flow* 23, 592–600 (2002).
- Voke, P. R., S. Gao and D. Leslie. "Large-eddy simulations of plane impinging jets." International Journal for Numerical Methods in Engineering 38, 489–507 (1995).
- Gao, S. and P. R. Voke. "Large-eddy simulation of turbulent heat transport in enclosed impinging jets." Int. J. Heat and Fluid Flow 16, 349–356 (1995).
- Olsson, M. and L. Fuchs. "Large eddy simulation of a forced semiconfined circular impinging jet." *Phys. Fluids* 10(2), 476–486 (1998).
- Cziesla, T., E. Tandogan and N. K. Mitra. "Large-eddy simulation of heat transfer from impinging slot jets." Numerical Heat Transfer, Part A 32, 1–17 (1997).
- Horiuti, K. "Large eddy simulation of turbulent channel flow by one-equation modeling." Journal of Physical Society of Japan 54(8), 2855–2865 (1985).
- Thuburn, J. "Mutidimensional flux-limited advection schemes." Journal of Computational physics 123, 74–83 (1996).
- 67. Pope, S. B. *Turbulent Flows* (Cambridge University Press, Cambridge, UK, 2000).
- Landreth, C. C. and R. J. Adrian. "Impingement of a low Reynolds number turbulent circular jet onto a flat plate at normal incidence." *Experiments in Fluids* 9, 74–84 (1990).
- Didden, N. and C. M. Ho. "Unsteady separation in a boundary layer produced by an impinging jet." J. of Fluid Mech. 160, 235–256 (1985).

- Thomas, B. G., X. Huang and R. C. Sussman. "Simulation of argon gas flow effects in a continuous slab caster." *Metallurgical Transactions B* 25B, 527–547 (1994).
- Thomas, B. and L. Zhang. "Mathematical modeling of fluid flow in continuous casting." *ISIJ International* 41(10), 1181–1193 (2001).
- Thomas, B. G., L. J. Mika and F. M. Najjar. "Simulation of fluid flow inside a continuous slab-casting machine." *Metallurgical Transactions B* 21B, 387–400 (1990).
- 73. Creech, D. T. Computational Modelling of Multiphase Turbulent Fluid Flow and Heat Transfer in the Continuous Slab Casting Mold. Master's thesis, University of Illinois at Urbana-Champaign (1999).
- 74. Offerman, C. "Affect on the quality and internal structure in continuously cast slabs by the metal flow in the mould." *Scandinavian Journal of Metallurgy* **10**(1).
- 75. Dauby, P. H., M. B. Assar and G. Lawson. "Voyage dans une lingotiere de coulee continue. mesures laser et electromagnetiques de l'hydrodynamique de l'acier (PIV and MFC measurements in a continuous caster mould. new tools to penetrate the caster black box)." *Revue de Metallurgie. Cahiers D'Informations Techniques* 98(4).
- 76. Thomas, B. G., Q. Yuan, S. Sivaramakrishnan, T. Shi, S. P. Vanka and M. B. Assar. "Comparison of four methods to evaluate fluid velocities in a continuous casting mold." *ISIJ International* 41(10).
- 77. Gupta, D. and A. K. Lahiri. "Water-modeling study of the surface disturbances in continuous slab caster." *Metallurgical and Materials Transactions B* **25B**(2).
- Gupta, D. and A. K. Lahiri. "A water model study of the flow asymmetry inside a continuous slab casting mold." *Metallurgical and Materials Transactions B* 27B(5).
- 79. Gupta, D., S. Chakraborty and A. K. Lahiri. "Asymmetry and oscillation of the fluid flow pattern in a continuous casting mould: a water model study." *ISIJ International* 37(7).
- Sobolewski, R. and H. D. J. "Water modeling of slab caster flow conditions." In 2nd Process Technology Conference Proceedings, Vol. 2, pp. 160–165 (Iron and Steel Scociety, Warrendale, PA, 1982).

- Huang, X., B. G. Thomas and F. M. Najjar. "Modeling superheat removal during continuous casting of steel slabs." *Metallurgical Transactions B* 23B, 339–356 (1992).
- Thomas, B. G. and F. M. Najjar. "Finite element modelling of turbulent fluid flow and heat transfer in continuous casting." *Appl. Math. Modelling* 15, 226–243 (1991).
- 83. Seyedein, S. H. and M. Hasan. "Three-dimensional simulation of coupled turbulent flow and macroscopic solidification heat transfer for continuous slab casters." *International Journal of Heat and Mass Transfer* 40(18).
- Seyedein, S. H. and M. Hasan. "3-D numerical prediction of turbulent flow, heat transfer and solidification in a continuous slab caster for steel." *Canadian Metallurgical Quarterly* 37(3-4).
- 85. "CFX V4.2." AEA Technology. (1997).
- 86. Thomas, B. G., R. J. O'Malley, T. Shi, Y. Meng, D. Creech and D. T. Stone. "Validation of fluid flow and solidification simulation of a continuous thin-slab caster." In *Proceedings of Modelling of Casting, Welding, and Advanced Solidification Processes IX*, pp. 769–776 (The Minerals, Metals and Materials Scociety, Archen, Germany, 2000).
- 87. Thomas, B. G., R. J. O'Malley and D. T. Stone. "Measurement of temperature, solidification, and microstructure in a continuous cast thin slab." In *Proceedings* of Modelling of Casting, Welding, and Advanced Solidification Processes VIII, pp. 1185–1199 (The Minerals, Metals and Materials Scociety, Warrendale, PA, 1998).
- Yuan, Q., B. G. Thomas and S. P. Vanka. "Study of transient flow and particle transport in continuous steel caster molds: Part I. fluid flow." *Metallurgical and Materials Transactions B* (in review, 2003).
- Yuan, Q., B. G. Thomas and S. P. Vanka. "Study of transient flow and particle transport in continuous steel caster molds: Part II. particle transport." *Metallurgical and Materials Transactions B* (accepted, 2003).
- 90. Shi, T. Effect of Argon Injection on Fluid Flow and Heat Transfer in the Continuous Slab Casting Mold. Master's thesis, University of Illinois at Urbana-Champaign (2001).

 Jimbo, I. and A. W. Cramb. "The density of liquid iron-carbon alloys." Metallurgical Transactions B 24B, 5–10 (1993).