ARGON BUBBLE TRANSPORT AND CAPTURE IN CONTINUOUS CASTING WITH AN EXTERNAL MAGNETIC FIELD USING GPU-BASED LARGE EDDY SIMULATIONS

BY

KAI JIN

DISSERTATION

Submitted in partial fulfillment of the requirements for the degree of Doctor of Philosophy in Mechanical Engineering in the Graduate College of the University of Illinois at Urbana-Champaign, 2016

Urbana, Illinois

Doctoral Committee:

Professor Brian G. Thomas, Co-Chair and Co-Director of Research
Professor Surya P. Vanka, Co-Chair and Co-Director of Research
Professor David N. Ruzic
Assistant Professor Caleb Brooks
Assistant Professor Kyle C. Smith
Abstract

Continuous casting produces over 95% of steel in the world today, hence even small improvements to this important industrial process can have large economic impact. In the continuous casting of steel process, argon gas is usually injected at the slide gate or stopper rod to prevent clogging, but entrapped bubbles may cause defects in the final product. Many defects in this process are related to the transient fluid flow in the mold region of the caster. Electromagnetic braking (EMBr) device is often used at high casting speed to modify the mold flow, reduce the surface velocity and fluctuation. This work studies the physics in continuous casting process including effects of EMBr on the motion of fluid flow in the mold region, and transport and capture of bubbles in the solidification processes. A computational effective Reynolds-averaged Navier-Stokes (RANS) model and a high fidelity Large Eddy Simulation (LES) model are used to understand the motion of the molten steel flow. A general purpose multi-GPU Navier-Stokes solver, CUFLOW, is developed. A Coherent-Structure Smagorinsky LES model is implemented to model the turbulent flow. A two-way coupled Lagrangian particle tracking model is added to track the motion of argon bubbles. A particle/bubble capture model based on force balance at dendrite tips is validated and used to study the capture of argon bubbles by the solidifying steel shell. To investigate the effects of EMBr on the turbulent molten steel flow and bubble transport, an electrical potential method is implemented to solve the magnetohydrodynamics equations. Volume of Fluid (VOF) simulations are carried out to understand the additional resistance force on moving argon bubbles caused by adding transverse magnetic field. A modified drag coefficient is extrapolated from the results and used in the two-way coupled Eulerian-Lagrangian model to predict the argon bubble transport in a caster with EMBr. A hook capture model is developed to understand the effects of hooks on argon bubble capture.
To My Parents and My Wife.
Acknowledgments

These projects would not have been possible without the support of many people. Many thanks to my advisors, Dr. Brian G. Thomas and Dr. Surya P. Vanka, who guided me during the research and read my numerous revisions. Also thanks to my committee members, Dr. David Ruzic, Dr. Caleb Brooks, and Dr. Kyle Smith, who offered guidance and support. Thanks to the Continuous Casting Consortium members (ABB, AK Steel, ArcelorMittal, Baosteel, JFE Steel Corp., Magnesita Refractories, Nippon Steel and Sumitomo Metal Corp., Nucor Steel, Posco, SSAB, ANSYS/Fluent) and U.S. National Science Foundation Grant (CMMI-11-30882) for the founding support. Thanks to Baosteel (China) for providing valuable measurement data. Thanks to BlueWaters / National Center for Supercomputing Applications (NCSA) at UIUC for providing allocation on BlueWaters. Thanks to NVIDIA for providing the GPU cards through the NVIDIA Professor Partnership program. I would like to give special thanks to my wife, Yan, for being incredibly supportive and patient during the course of this Ph.D. program. And finally, thanks to my parents and numerous friends who endured this long process with me, always offering support and love.
# Table of Contents

List of Tables ........................................................................................................ viii
List of Figures .......................................................................................................... ix
List of Abbreviations ............................................................................................... xiv
List of Symbols ......................................................................................................... xvi

Chapter 1 Introduction .............................................................................................. 1
  1.1 Continuous Casting of Steel .............................................................................. 1
  1.2 Objectives of the research .............................................................................. 2
  1.3 Physical Phenomena and Literature Review .................................................... 2
    1.3.1 Effect of EMBr ......................................................................................... 3
    1.3.2 Effect of Argon Gas Injection .................................................................. 3
    1.3.3 Inclusion Transport and Capture .............................................................. 4
    1.3.4 Bubble and Particle Capture at Solidification Front ............................... 5
  1.4 References ........................................................................................................ 6

Chapter 2 Mathematic Models and CUFLOW ........................................................ 9
  2.1 Navier-Stokes Equations and Fraction-Step Method ........................................ 9
  2.2 Large Eddy Simulation Model ......................................................................... 11
  2.3 Magnetohydrodynamics Model ....................................................................... 12
  2.4 Two-way Coupled Eulerian-Lagrangian Particle Tracking ............................... 13
    2.4.1 Particle Motion Equations and Forces on Particle ................................. 13
    2.4.2 Effects of Ferro-static Pressure on Bubble Size ....................................... 15
    2.4.3 Breakup and Coalescence of Argon Bubbles .......................................... 16
  2.5 Volume of Fluid (VOF) Model ........................................................................ 17
  2.6 In-house Code CUFLow and GPU Computing ............................................... 19
    2.6.1 GPU and Its Architecture ...................................................................... 19
    2.6.2 Programing on GPU and GPU Speedup ................................................ 20
    2.6.3 Multi-GPU Implementation and Speedup .............................................. 24
  2.7 References ........................................................................................................ 27

Chapter 3 Modeling and Measurements of Multiphase Flow and Bubble Entrapment 30
  3.1 Introduction ..................................................................................................... 30
  3.2 Plant Measurements ......................................................................................... 33
  3.3 Computational Models and Solution Procedure ............................................. 35
    3.3.1 Initial Flow Model .................................................................................... 35
    3.3.2 Bubble Size Distribution Model ............................................................... 37
    3.3.3 Two-phase Flow Model ........................................................................... 38
    3.3.4 Bubble Tracking and Capture Model ...................................................... 40
    3.3.5 Model to Predict Sample Observations ................................................ 42
Chapter 6  Single Argon Bubble Rise in Liquid Steel under External Magnetic Field

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1</td>
<td>Introduction</td>
<td>122</td>
</tr>
<tr>
<td>6.2</td>
<td>Governing Equations and Solution Procedure</td>
<td>129</td>
</tr>
<tr>
<td>6.3</td>
<td>Code Validation and Grid Independence Study</td>
<td>133</td>
</tr>
<tr>
<td>6.3.1</td>
<td>Validation 1. - Rise of an air bubble in water</td>
<td>133</td>
</tr>
<tr>
<td>6.3.2</td>
<td>Validation 2. - argon bubble rise in GaInSn</td>
<td>135</td>
</tr>
<tr>
<td>6.3.3</td>
<td>Validation 3. - N₂ bubble rise in mercury with horizontal magnetic field</td>
<td>137</td>
</tr>
<tr>
<td>6.4</td>
<td>Results and Discussions</td>
<td>138</td>
</tr>
<tr>
<td>6.4.1</td>
<td>Bubble rise velocity and deformation</td>
<td>139</td>
</tr>
<tr>
<td>6.4.2</td>
<td>Pressure and velocity fields and Lorentz force distribution</td>
<td>149</td>
</tr>
<tr>
<td>6.4.3</td>
<td>Flow structures behind the bubble</td>
<td>154</td>
</tr>
<tr>
<td>6.4.4</td>
<td>Modified drag coefficient</td>
<td>157</td>
</tr>
<tr>
<td>6.5</td>
<td>Summary</td>
<td>159</td>
</tr>
<tr>
<td>6.6</td>
<td>References</td>
<td>160</td>
</tr>
</tbody>
</table>

Chapter 7  Effect of EMBr on Ar Bubble Capture using LES Model

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.1</td>
<td>Introduction</td>
<td>164</td>
</tr>
<tr>
<td>7.2</td>
<td>Computational Models and Numerical Methods</td>
<td>166</td>
</tr>
<tr>
<td>7.2.1</td>
<td>Governing Equations for the Fluid Flow</td>
<td>168</td>
</tr>
<tr>
<td>7.2.2</td>
<td>Lagrangian Model for Argon Bubbles</td>
<td>169</td>
</tr>
<tr>
<td>7.2.3</td>
<td>Equations for Lorentz Forces</td>
<td>175</td>
</tr>
<tr>
<td>7.2.4</td>
<td>The Solver</td>
<td>175</td>
</tr>
<tr>
<td>7.3</td>
<td>Results and Discussion</td>
<td>176</td>
</tr>
<tr>
<td>7.3.1</td>
<td>Motion of liquid steel</td>
<td>176</td>
</tr>
<tr>
<td>7.3.2</td>
<td>Transport of argon bubbles</td>
<td>177</td>
</tr>
<tr>
<td>7.3.3</td>
<td>Capture and removal of argon bubbles</td>
<td>181</td>
</tr>
<tr>
<td>7.3.4</td>
<td>Compare capture rate and fraction with previous RANS model</td>
<td>182</td>
</tr>
<tr>
<td>7.3.5</td>
<td>Effects of EMBr on flow field and argon bubble transport</td>
<td>186</td>
</tr>
<tr>
<td>7.3.6</td>
<td>Effects of EMBr on argon bubble capture and removal</td>
<td>187</td>
</tr>
<tr>
<td>7.4</td>
<td>Conclusions</td>
<td>190</td>
</tr>
<tr>
<td>7.5</td>
<td>References</td>
<td>191</td>
</tr>
</tbody>
</table>

Chapter 8  Effect of Hooks on Bubble Capture

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.1</td>
<td>Introduction</td>
<td>196</td>
</tr>
<tr>
<td>8.2</td>
<td>Hook Capture Mechanism</td>
<td>196</td>
</tr>
<tr>
<td>8.3</td>
<td>Results and Discussion</td>
<td>198</td>
</tr>
<tr>
<td>8.3.1</td>
<td>Bubbles Captured Using 6 mm Hook Depth</td>
<td>198</td>
</tr>
<tr>
<td>8.3.2</td>
<td>Bubbles Captured Using 3 mm Hook Depth</td>
<td>199</td>
</tr>
<tr>
<td>8.3.3</td>
<td>Capture Locations and Number of Small Bubbles</td>
<td>199</td>
</tr>
<tr>
<td>8.4</td>
<td>Compare with Previous Prediction and Plant Measurements</td>
<td>201</td>
</tr>
<tr>
<td>8.5</td>
<td>Conclusions</td>
<td>202</td>
</tr>
</tbody>
</table>

Chapter 9  Conclusions and Recommendations

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.1</td>
<td>Effects of Submergence Depth</td>
<td>203</td>
</tr>
<tr>
<td>9.2</td>
<td>Effects of Casting Speed</td>
<td>204</td>
</tr>
<tr>
<td>9.3</td>
<td>Effects of EMBr on Steel Flow when Argon Injection Can Be Ignored</td>
<td>204</td>
</tr>
<tr>
<td>9.4</td>
<td>Effects of EMBr on Argon Bubble Rise in Liquid Steel</td>
<td>205</td>
</tr>
<tr>
<td>9.5</td>
<td>Effects of Capture Criterion on Model Prediction</td>
<td>206</td>
</tr>
<tr>
<td>9.6</td>
<td>Effects of EMBr on the Capture of Argon Bubbles</td>
<td>206</td>
</tr>
<tr>
<td>9.7</td>
<td>Effect of Hook on the Capture of Argon Bubbles</td>
<td>207</td>
</tr>
</tbody>
</table>
List of Tables

2.1 Speed up using different CPU and GPU with problem size: \( N_x \times N_y \times N_z = 256 \times 256 \times 256 \)  23
2.2 Timing for different problem sizes (Best CPU Speedup vs Best GPU Speedup)  23

3.1 Process parameters  34
3.2 Capture fractions of 1 mm bubbles  55

4.1 List of all simulations  69
4.2 \( Re = 400 \) u velocity on vertical center line \( x = 0.5 \)  71
4.3 \( Re = 400 \) v velocities on horizontal line \( y = 0.5 \)  74
4.4 \( Re = 2000 \) u velocity on vertical center line \( x = 0.5 \)  76
4.5 \( Re = 2000 \) v velocity on horizontal line \( y = 0.5 \)  76
4.6 \( Re = 3200 \) u velocity on vertical center line \( x = 0.5 \)  79
4.7 \( Re = 3200 \) v velocity on horizontal line \( y = 0.5 \)  80
4.8 \((x,y)\) coordinates of the eye of the primary eddy in \( z = 0.5 \) plane  83

5.1 Process parameters (*: Casting conditions when plant measurements were carried out)  96
5.2 List of the eight LES simulations (*Note: measurements available for this condition)[39]  99
5.3 The standard deviations of surface level \( \sigma(h) \) (mm)  116

6.1 Summary of studies of bubble rise under external magnetic field  123
6.2 Properties of air and water, argon and steel  133
6.3 Dimensionless numbers of air-water and Ar-steel \((d = 3 \text{ mm})\)  133
6.4 Property of argon and GaInSn at \( T = 293\text{K} \)  136
6.5 Physical properties of \( N_2 \) and mercury at \( T \approx 300\text{K} \)  137
6.6 List of simulations \((\text{Ar bubble rising in liquid steel, } 1773 \text{ K, } Mo = 1.3 \times 10^{-12})\)  138
6.7 Effect of Stuart number \( N (Ha^2/Re) \) on drag coefficient \( C_D \)  158

7.1 Process parameters  168
# List of Figures

1.1 Continuous casting process ................................. 1

2.1 (a) Bubble diameter as a function of $z$  
(b) Bubble density as a function of $z$ .................. 16

2.2 Gas pockets in below the slide gate (left) and at the top of port (right) .......... 17

2.3 A Tesla K40 GPU made by Nvidia ............................ 20

2.4 Nvidia GK110 streaming multiprocessor (SMX) architecture ....................... 21

2.5 The red-black ordering (left) and finite difference stencil (right) ................ 22

2.6 The steps of computing the final temperature using red-black SOR on GPU ......... 22

2.7 (a) Speedup using different CPU and GPU with problem size: $N_x \times N_y \times N_z = 256 \times 256 \times 256$ 
and estimated price for the device.  
(b) the comparison of price and timing for different problem sizes .................... 24

2.8 Domain decomposition illustrated in 2D with 4 GPUs ................................. 24

2.9 Additional layers at the domain boundaries ............................................. 25

2.10 Data transfer between two nodes ......................................................... 25

2.11 The virtual machine setup ................................................................. 26

2.12 (a) Speedup of the multi-GPU CUFLOW (b) Stream traces in the center plane 
(c) Contours of the $v$ velocity in the center plane ....................................... 27

3.1 Slide gate configuration schematic ................................................. 33

3.2 Locations of samples and six layers examined on each ............................ 33

3.3 Example of large bubble ($d_p = 1.45$ mm) found in plant measurements .... 34

3.4 Mesh of the computational domain of the slide gate, SEN, and mold region .... 35

3.5 Flow chart of the entire solution procedure ........................................... 35

3.6 Computation of mass sink term for shell solidification ............................. 36

3.7 Ar bubble volume fraction distribution ................................................. 38

3.8 Flow chart for Ar bubble/particle capture criterion .................................. 41

3.9 A Bubble/Particle Touching 3 Dendrite Tips ......................................... 41

3.10 Determine the number of bubbles can be observed on surface .................... 43

3.11 Convert hole diameter observed on sample surface to true bubble diameter .... 43

3.12 Ar volume fraction and velocity field in symmetry plane of SEN ............... 44

3.13 Ar volume fraction and steel vector field at port outlet .......................... 45

3.14 Velocity magnitude and streamlines in middle plane ............................... 45

3.15 Velocity and Ar volume fraction near top surface ................................... 46

3.16 Small bubbles captured by WF-IR (left), WF-OR (middle) and NF (right) using simple capture criterion .................................................. 47

3.17 Small bubbles captured by WF-IR (left), WF-OR (middle) and NF (right) using advanced capture criterion ............................................. 47

3.18 Large bubbles ($d_p \geq 1mm$) captured by WF-IR (left), WF-OR (middle) and NF (right) using simple capture criterion .................................. 47

3.19 Large bubbles ($d_p \geq 1mm$) captured by WF-IR (left), WF-OR (middle) and NF (right) using advanced capture criterion .................................. 47
3.20 Bubbles escaped from top surface using simple (top) and advanced (bottom) capture criterion
3.21 Number of bubbles captured per 6 sec in entire caster .................................................. 48
3.22 Bubble capture fraction for different bubble diameters .................................................... 49
3.23 Sankey diagram of number of bubbles captured in different zones for (a) large bubbles (b) medium bubbles and (c) small bubbles .................................................. 50
3.24 Capture rate for small, medium and large bubbles in different zones ............................. 51
3.25 Predicted number of bubbles captured per cm² at (a) center region, (b) quarter region and (c) NF region .................................................. 52
3.26 Compare centerline velocity on top surface with plant measurement ............................... 53
3.27 Predicted number of bubbles captured in each layer of center sample (a) and quarter sample (b) Predicted average bubble diameter in center sample (c) and quarter sample (d) ...... 53
3.28 Predicted (a) number of bubbles captured in each layer of NF sample and (b) corresponding average bubble diameter .................................................. 54

4.1 Comparison of velocity profiles in symmetry plane of a lid-driven cube, Re =1000 .......... 66
4.2 Validation result of 2D cavity MHD flow at Re = 5000 and N = 5 ................................. 67
4.3 Grid independency study, Re = 3200 and N = 0.25 in symmetry plane (z=0.5) .............. 67
4.4 The lid driven cavity with an external magnetic field ....................................................... 68
4.5 Velocity magnitude contour at Re=400 and N=0.0 on (a) z=0.05 (b) z=0.1 (c) z=0.3 and (d) z=0.5 .................................................................................. 70
4.6 Velocity magnitude contour at Re=400 and N=1.0 on (a) z=0.05 (b) z=0.1 (c) z=0.2 and (d) z=0.5 .................................................................................. 71
4.7 Velocity magnitude contour at Re=400 and N=2.0 on (a) z=0.05 (b) z=0.1 (c) z=0.3 and (d) z=0.5 .................................................................................. 71
4.8 Streamlines in z = 0.5 plane, Re = 400 for different N ...................................................... 72
4.9 (a) u, (b) v and (c) w velocities on spanwise centerline .................................................. 73
4.10 Centerline velocities in z = 0.5 plane, Re = 400 for different magnetic fields .............. 73
4.11 Streamlines in z = 0.5 plane, Re = 2000 for different N ................................................. 77
4.12 Centerline velocity in z = 0.5 plane, Re = 2000 .................................................. 78
4.13 Velocity vector in middle x plane for Re = 2000 .................................................. 78
4.14 Streamlines in z = 0.5 plane, Re = 3200 for different N (from left to right, top to bottom): N = 0, 0.04, 0.0625, 0.09, 0.25, 0.5, 1.0, 2.0 .............................................. 81
4.15 Centerline velocity in z = 0.5 plane, Re = 3200 .................................................. 82
4.16 Velocity vector in middle x plane for Re = 3200 .................................................. 82
4.17 Streamlines in z = 0.5 plane, Re = 5000 for N = 0.25 and N = 0.50 ............................ 83

5.1 Shell thickness profile at two casting speeds used in the computational model ............... 94
5.2 Schematic of grid near solid shell showing a cell with non-zero mass sink terms ............ 94
5.3 Computational domain (cells in the solid shell region are hidden) .................................. 97
5.4 Slide gate showing asymmetric flow in upper nozzle, 80% open with casting speed of 1.8 m/min .................................................. 97
5.5 Measured magnetic field showing measurement points (dots) and interpolated (lines) used in model ................................................................................. 98
5.6 Location of important surfaces ...................................................................................... 99
5.7 Velocity measurement device ...................................................................................... 100
5.8 Comparison of predicted surface velocity (centerplane, quarter mold, 1 cm below top surface) with plant measurements .................................................. 100
5.9 (a) Contours of time-averaged $u_x$ velocity in centerplane of SEN; (b) snapshot of transient $u_z$ at $t = 64.8s$ .................................................. 101
5.10 (a) time-averaged $x$ velocity, $u_x < 0$, flow enter mold; $u_x > 0$, back flow reversal going into port (b) a snapshot of $u_y$ at $t = 64.8s$ .................................................. 102
5.11 Steel shell, SEN and iso-surface of constant $u_x = \pm 0.7$ m/s at $t = 64.8$ s .................. 103
5.12 Contours of time-average velocity magnitude at (a) 1 cm below top surface and (b) in mold centerplane .................................................. 103
5.13 Swirl at bottom of SEN (low casting speed of 1.3 m/min) .................................................. 104
5.14 Top surface velocity and swirl directions in SEN port at low casting speed ........................................ 105
5.15 Swirl directions in bottom of SEN for different submergence depths (no EMBr) ........................................ 105
5.16 Effect of submergence depth on time-averaged velocity and swirl in port (no EMBr) ................................. 106
5.17 Effect of submergence depth on velocity in mold. (a) and (b) show flow at 1 cm below top surface; (c) and (d) show flow in mold centerplane. (Vectors shown at only 1/30 of all grid points.) .................................................. 107
5.18 Snapshot of instantaneous velocity at top surface (no EMBr) ................................................................. 108
5.19 Time-averaged flow in SEN symmetry plane (x = 0) under different magnetic field. Contour of \( \bar{u}_x \) .................................................................................................................................................. 108
5.20 Contour of \( \bar{u}_z \) in SEN centerplane. (a) No EMBr; (b) bottom coil current 850 A; (c) top coil current 400 A and bottom coil current 850 A; (d) both coils have current of 850 A ........................................... 109
5.21 Time averaged profile of \( \bar{u}_z \) in SEN at \( z = 1.5 \) m and \( y = 0 \) ........................................................................ 110
5.22 Velocity at bottom of the SEN. Positive \( y \) velocity means flow exiting port (a) No EMBr; (b) bottom coil current 850 A; (c) top coil current 400 A and bottom coil current 850 A; (d) both coils have current of 850 A .......................................................... 111
5.23 Contours of time-averaged x-velocity in port (positive \( x \) velocity means back flow) for \( d_{sub} = 170 \) mm with different EMBr: (a) without EMBr (b) bottom coil current 850 A; (c) top coil current 400 A and bottom coil current 850 A; (d) both coils have current of 850 A ......................................................... 111
5.24 Effect of EMBr on swirl rotation at bottom of SEN (170mm submergence) ................................................. 112
5.25 Time averaged velocity magnitude \( |\bar{u}| \) in centerline plane of the mold ......................................................... 113
5.26 Contours of time-averaged velocity magnitude at 1 cm below top surface (a) bottom coil current 850 A; (b) top coil current 400 A and bottom coil current 850 A; (c) both coils have current of 850 A .......................................................... 113
5.27 Mean \( x \)-velocity across center line 1cm below top surface. (Legend shows current (A) in top and bottom coils) .................................................................................................................................................. 114
5.28 (a) Top view showing points where pressure data were obtained on centerplane. (b) Predicted surface level profiles for cases with submergence depth 170 mm. (Legend shows current (A) in top and bottom coils) ........................................................................................................................................ 115
5.29 Predicted surface level profile for cases with submergence depth 200 mm. (Legend shows current (A) in top and bottom coils) ........................................................................................................................................ 115
5.30 Surface level history at (a) point P7, (b) point P8, and (c) point P9 with different magnetic field strengths (see Fig. 5.28 for locations. Legend shows current in top and bottom coils in Amps). ........................................................................................................................................ 116
5.31 Predicted TKE at the middle plane for cases with submergence depth 170 mm and varies magnetic field strengths a) without EMBr (b) bottom coil current 850 A; (c) top coil 400 A and bottom coil 850 A; (d) both coils 850 A .......................................................... 117
6.1 Computational domain and the initial bubble location .................................................................................. 130
6.2 A flow chart shows basic solution procedures ............................................................................................... 132
6.3 \( E_o \) vs. predicted \( Re_f \) on the Grace diagram. Bubble shape: scouw - spherical cap with open unsteady wake; scsw - spherical cap with closed steady wake; swu - skirt with wavy unsteady skirt; sss - skirt with smooth steady skirt ......................................................................................................................................... 134
6.4 (a) Comparison of predicted terminal velocities of bubbles of different size (b) Predicted shape of a 2 mm bubble after the bubble travels ~6.7 mm from release ........................................................................................................ 135
6.5 (a) History of rise velocity of 2.5 mm Ar bubble rising in GaInSn (b) Comparison of predicted terminal velocity and available data for Ar bubbles in GaInSn .................................................................................................................................. 136
6.6 Rise velocity of a \( d = 5.6 \) (\( E_o = 8.5 \)) mm \( N_2 \) bubble rise in mercury subject to a horizontal magnetic filed of 0.5 T \( (N = 0.45 \) ........................................................................................................................................ 137
6.7 (a) Rise velocity of 3 mm Ar bubble and (b) Rise velocity of 7 mm Ar bubble ........................................................................................................................................ 139
6.8 Bubble rise velocity and the associated top-views of the 3 mm bubble at different times (indicated as black dots on the line plot) for \( B = 0 \) T. ........................................................................................................................................ 140
6.9 Bubble shape at different time \( t^* \), \( d = 3 \) mm, \( B = 0 \) T ........................................................................................................................................ 141
6.10 Bubble rise velocity and the associated top-view of the 3 mm bubble at different times indicated as red triangle on the line plot with $B = 0.2$ T. ........................................... 141
6.11 Bubble shape at different time $t^*$, $d = 3$ mm, $B = 0.2$ T ................................. 142
6.12 Rise velocity and different views of the 3 mm bubble with $B = 0.5$ T. .......................... 142
6.13 Bubble shape at different time $t^*$, $d = 7$ mm, without magnetic field .................... 143
6.14 Bubble rise velocity and the associated top-views of the 7 mm bubble at different times (shown as black dots on the line plot) $B = 0$ T. .................................................. 144
6.15 Range of the shape oscillation frequency $f_{z,0}$ .................................................................. 145
6.16 Front views of the 7 mm bubble at different time $t^*$, without magnetic field .......... 146
6.17 Bubble rise velocity and the associated top-view of the 7 mm bubble at different times (shown as red triangle on the line plot) with $B = 0.2$ T. ........................................... 147
6.18 Front views of the 7 mm bubble at different time $t^*$, $B = 0.2$ T ................................. 148
6.19 Bubble rise velocity and the associated top-view of the 7 mm bubble at different time (shown as blue square on the line plot) with $B = 0.5$ T. ........................................... 148
6.20 Contours of $p^*$ at $t^* = 2.5$, $d = 7$ mm, at middle plane $x/d = 3$ .......................... 149
6.21 Contours of $u^*$ at $t^* = 2.5$, $d = 7$ mm, middle plane $x/d = 3$ ............................... 150
6.22 Contours of $w^*$ at $t^* = 8.75$, $d = 7$ mm, middle plane $x/d = 3$ ......................... 150
6.23 Contours of $u^*$ in central $x$-$z$ plane (top) and contours of $v^*$ in central $y$-$z$ plane (bottom) for the three different magnetic field strengths at $t^* = 2.5$ ............................... 151
6.24 Contour plots of $F_y^*$ and $F_z^*$ at $t^* = 2.5$ and direction of Lorentz force, for bubble $d = 7$ mm at middle plane $x/d = 3$ ................................................................. 152
6.25 (a) Vectors of Lorentz force in $y$-$z$ plane and $x$-$z$ plane and eight isosurfaces of constant $F_y^*$ (b) Contours of $F_y^*$ at top and bottom half of the rising 7 mm bubble, $B = 0.5$ T and $t^* = 8.75$ .................................................. 153
6.26 Front and side views of the isosurfaces of $|\omega^*| = 3$ at $t^* = 5.75$ under different magnetic field strengths, original bubble diameter $d = 3$ mm ................................. 154
6.27 Front and side views of the bubble (green), isosurfaces of $\omega^*_x = 1$ (bright yellow) and $\omega^*_x = -1$ (dark blue) at $t^* = 5.75$ for 3 mm bubble .................................................. 155
6.28 Top views of the bubble (green), isosurfaces of $\omega^*_x = 1$ (bright yellow) and $\omega^*_x = -1$ (dark blue) at $t^* = 5.75$ of 3 mm bubble for $B = 0T$, $B = 0.2T$ and $B = 0.5T$ (from left to right) .................................................. 156
6.29 Front and side views of the bubble (green), isosurfaces of $|\omega^*| = 4$ (pink color) at $t^* = 9$ for 7 mm bubble with $B = 0T$, $B = 0.2T$, and $B = 0.5T$ .................................................. 156
6.30 Front and side views of the bubble (green), isosurfaces of $\omega^*_x = 1$ (bright yellow) and $\omega^*_x = -1$ (dark blue) at $t^* = 9$ for 7 mm bubble with $B = 0T$, $B = 0.2T$, and $B = 0.5T$ .................................................. 157
6.31 Effect of $N$ on the drag coefficient ................................................................................. 159
7.1 Computational domain ..................................................................................................... 167
7.2 Slide gate position, 70% open at $V_c = 1.5$ m/min ......................................................... 167
7.3 Shell profile ..................................................................................................................... 168
7.4 Number of bubbles injected in every 0.01 second .......................................................... 170
7.5 (a) Predicted rise velocities of air bubble in water and argon bubble in steel (b) Predicted bubble rise velocity on Grace Diagram. ......................................................... 172
7.6 An argon bubble bounces back after hit SEN wall .......................................................... 174
7.7 Flow chart shows solution procedure .............................................................................. 176
7.8 Time-averaged flow field (a) inside SEN, (b) at the centerplane of mold and (c) 1 cm below top surface, without EMBr .................................................................. 177
7.9 Bubble inside the SEN and near the symmetry plane $x = 0$. Visualized all bubbles with $|x_p| \leq 2$ mm ......................................................................................... 178
7.10 Contours of $y$-velocity in the symmetry plane at SEN bottom at (a) $t = 23.5$ s and (b) $t = 24$s. All bubbles are projected into the symmetry plane and are drawn to scale. ................................. 179
7.11 Contours of $u_x$ velocity and bubbles near the middle plane $(y = 0$ mm; showing bubbles with: -2 mm $< y_p < 2$ mm) ........................................................................ 179
7.12 Contour of $u_x$ velocity and bubbles in the middle plane $(y = 0$ mm) near port exit. .......... 180
7.13 Active bubbles in the caster projected to the middle plane $(y = 0$). ................................ 181
7.14 Injection rate, removal rate (particles escaped from top surface) and capture rate (captured by shells) ................................................................. 181
7.15 Total number of bubbles has been injected into the caster and remaining bubbles in the caster at \( t = 32s \) ................................................................. 182
7.16 Compare the predicted capture fraction with previous RANS model ................................................................. 183
7.17 Compare the predicted capture rate with previous RANS model ................................................................. 183
7.18 Compare predicted average bubble diameter on NF with previous RANS model and measurements ................................................................. 184
7.19 Compare predicted average bubble diameter on WF-IR with previous RANS model and measurements ................................................................. 184
7.20 Bubbles captured by the solidifying shell during the simulation (without EMBr): (a) WF-OR; (b) WF-IR; (c) NF. ................................................................. 185
7.21 Bubbles escaped from the top surface. Sizes of the bubbles are not drawn to scale. ................................................................. 186
7.22 Time-averaged flow field (a) inside SEN, (b) at the centerplane of mold and (c) 1 cm below top surface, with EMBr. ................................................................. 186
7.23 Side views of bubbles inside the caster at \( t = 25s \): (a) EMBr off; (b) EMBr on. Showing all bubbles and bubbles are drawn to scale. ................................................................. 187
7.24 Bubbles captured by the solidifying shell during the simulation (with EMBr): (a) WF-OR; (b) WF-IR; (c) NF. ................................................................. 188
7.25 Effect of EMBr on bubble size distribution inside caster ................................................................. 188
7.26 Effect of EMBr on predicted argon bubble capture rate vs time ................................................................. 189
7.27 Compare the predicted (a) bubble capture fraction and (b) capture rate with/without EMBr 190
8.1 Modeled hook shape and the fate of a bubble/particle after it enters hook zone ................................................................. 196
8.2 (a) Flow chart of hook capture mechanism (b) Illustration of reset \( t_c \) ................................................................. 197
8.3 (a) Small bubbles captured by inner-radius hook (hook depth 6 mm); (b) Large bubbles captured by inner-radius hook (hook depth 6 mm). Legend shows the bubble diameter (\( \mu m \)). ................................................................. 198
8.4 (a) Small bubbles captured by outer-radius hook (hook depth 6 mm); (b) Large bubbles captured by outer-radius hook (hook depth 6 mm). Legend shows the bubble diameter (\( \mu m \)). ................................................................. 198
8.5 (a) Small bubbles captured by inner-radius hook (hook depth 3 mm); (b) Large bubbles captured by inner-radius hook (hook depth 3 mm). Legend shows the bubble diameter (\( \mu m \)). ................................................................. 199
8.6 (a) Small bubbles captured by outer-radius hook (hook depth 3 mm); (b) Large bubbles captured by outer-radius hook (hook depth 3 mm). Legend shows the bubble diameter (\( \mu m \)). ................................................................. 200
8.7 Small bubbles captured by WF-IR (left), WF-OR (middle) and NF (right) using 3 mm hook depth. ................................................................. 200
8.8 Small bubbles captured by WF-IR (left), WF-OR (middle) and NF (right) using 6 mm hook depth. ................................................................. 201
8.9 Predicted number of bubbles captured per sample layer at (a) inner radius center region, (b) inner radius quarter region and (c) NF region ................................................................. 202
8.10 Predicted average bubble diameter in each sample layer at (a) inner radius center region, (b) inner radius quarter region and (c) NF region sample ................................................................. 202
# List of Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AC</td>
<td>Alternating Current</td>
</tr>
<tr>
<td>AMR</td>
<td>Adaptive Mesh Refinement</td>
</tr>
<tr>
<td>BW</td>
<td>Blue Waters Super Computer</td>
</tr>
<tr>
<td>CC</td>
<td>Continuous Casting</td>
</tr>
<tr>
<td>CPU</td>
<td>Central Processing Unit</td>
</tr>
<tr>
<td>CSF</td>
<td>Continuum Surface Force</td>
</tr>
<tr>
<td>CSM</td>
<td>Coherent-Structure Smagorinsky Model</td>
</tr>
<tr>
<td>CUDA</td>
<td>Compute Unified Device Architecture</td>
</tr>
<tr>
<td>DC</td>
<td>Direct Current</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>DPM</td>
<td>Discrete Phase Model</td>
</tr>
<tr>
<td>DRW</td>
<td>Discrete Random Walk</td>
</tr>
<tr>
<td>EMBr</td>
<td>Electromagnetic Braking</td>
</tr>
<tr>
<td>FTM</td>
<td>Front Tracking Method</td>
</tr>
<tr>
<td>GFM</td>
<td>Ghost Fluid Method</td>
</tr>
<tr>
<td>GPU</td>
<td>Graphics Processing Unit</td>
</tr>
<tr>
<td>HSMAC</td>
<td>Highly Simplified Marker and Cell</td>
</tr>
<tr>
<td>IBM</td>
<td>Immersed Boundary Method</td>
</tr>
<tr>
<td>IR</td>
<td>Inner Radius</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>LI</td>
<td>Left Inner Halo Layer</td>
</tr>
<tr>
<td>LO</td>
<td>Left Outer Halo Layer</td>
</tr>
<tr>
<td>MHD</td>
<td>Magnetohydrodynamics</td>
</tr>
<tr>
<td>MPI</td>
<td>Message Passing Interface</td>
</tr>
<tr>
<td>MUSIG</td>
<td>Multiple Size Group Model</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>-------------</td>
</tr>
<tr>
<td>NF</td>
<td>Narrow Face</td>
</tr>
<tr>
<td>OpenMP</td>
<td>Open Multi-Processing</td>
</tr>
<tr>
<td>OR</td>
<td>Outer Radius</td>
</tr>
<tr>
<td>PBM</td>
<td>Pressure Boundary Method</td>
</tr>
<tr>
<td>PDAS</td>
<td>Primary Dendrite Arm Spacing</td>
</tr>
<tr>
<td>PIV</td>
<td>Particle Image Velocimetry</td>
</tr>
<tr>
<td>PPE</td>
<td>Pressure-Poisson Equation</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-averaged Navier-Stokes</td>
</tr>
<tr>
<td>RDMA</td>
<td>Remote Direct Memory Access</td>
</tr>
<tr>
<td>RI</td>
<td>Right Inner Halo Layer</td>
</tr>
<tr>
<td>RMS</td>
<td>Root-Mean-Square</td>
</tr>
<tr>
<td>RO</td>
<td>Right Outer Halo Layer</td>
</tr>
<tr>
<td>SEN</td>
<td>Submerged Entry Nozzle</td>
</tr>
<tr>
<td>SGS</td>
<td>Sub-Grid Scale</td>
</tr>
<tr>
<td>SIMT</td>
<td>Single-Instruction and Multiple-Thread</td>
</tr>
<tr>
<td>SMX</td>
<td>Streaming Multiprocessors</td>
</tr>
<tr>
<td>SOR</td>
<td>Successive Over Relaxation</td>
</tr>
<tr>
<td>SSF</td>
<td>Sharp Surface Force</td>
</tr>
<tr>
<td>TFLOPS</td>
<td>Trillion Floating Point Operations Per Second</td>
</tr>
<tr>
<td>TKE</td>
<td>Turbulence Kinetic Energy</td>
</tr>
<tr>
<td>UDF</td>
<td>User Defined Functions</td>
</tr>
<tr>
<td>UDV</td>
<td>Ultrasound Doppler Velocimetry</td>
</tr>
<tr>
<td>UTN</td>
<td>Upper Tundish Nozzle</td>
</tr>
<tr>
<td>VOF</td>
<td>Volume Of Fluid</td>
</tr>
<tr>
<td>WF</td>
<td>Wide Face</td>
</tr>
</tbody>
</table>
## List of Symbols

Unless otherwise stated, the definitions of the symbols used in this work are listed:

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$d_p$</td>
<td>Bubble diameter</td>
</tr>
<tr>
<td>$d_{sub}$</td>
<td>Submergence depth of the SEN</td>
</tr>
<tr>
<td>$g$</td>
<td>The standard acceleration due to gravity</td>
</tr>
<tr>
<td>$h$</td>
<td>Surface level of the steel-slag interface</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>$m_p$</td>
<td>Mass of a single particle or bubble</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
</tr>
<tr>
<td>$u$</td>
<td>Velocity of the base fluid (liquid steel)</td>
</tr>
<tr>
<td>$u_p$</td>
<td>Velocity of the dispersed phase (argon bubble)</td>
</tr>
<tr>
<td>$x_p$</td>
<td>Bubble location</td>
</tr>
<tr>
<td>$B$</td>
<td>Magnetic field strength</td>
</tr>
<tr>
<td>$C_{CSM}$</td>
<td>A model constant in the CSM model</td>
</tr>
<tr>
<td>$C_D$</td>
<td>Drag coefficient</td>
</tr>
<tr>
<td>$C_L$</td>
<td>Lift force coefficient</td>
</tr>
<tr>
<td>$C_r$</td>
<td>Confinement ratio</td>
</tr>
<tr>
<td>$Eo$</td>
<td>Eötvös number</td>
</tr>
<tr>
<td>$F_L$</td>
<td>Lorentz force</td>
</tr>
<tr>
<td>$F_{pB}$</td>
<td>Net buoyancy force acting on a bubble</td>
</tr>
<tr>
<td>$F_{pD}$</td>
<td>Drag force acting on a bubble</td>
</tr>
<tr>
<td>$F_{pL}$</td>
<td>Lift force acting on a bubble</td>
</tr>
<tr>
<td>$F_{pP}$</td>
<td>Pressure gradient force acting on a bubble</td>
</tr>
<tr>
<td>$F_{pT}$</td>
<td>Total force acting on a bubble</td>
</tr>
<tr>
<td>$F_{pV}$</td>
<td>Virtual mass force (added mass force) acting on a bubble</td>
</tr>
<tr>
<td>$Ha$</td>
<td>Hartmann number</td>
</tr>
</tbody>
</table>
\( J \)  Current density

\( Mo \)  Morton number

\( N \)  Stuart number (magnetic interaction number)

\( Re \)  Reynolds number

\( Re_{m} \)  Magnetic Reynolds number

\( Re_{p} \)  Particle or bubble Reynolds number

\( S \)  Source terms

\( V_{c} \)  Casting speed

\( V_{\text{cell}} \)  Volume of a finite volume cell

\( We \)  Weber number

\( \alpha \)  Volume fraction of the discrete phase (argon gas)

\( \gamma \)  Surface tension

\( \epsilon \)  Rate of dissipation of the turbulence kinetic energy

\( \kappa \)  Mean interface curvature

\( \mu \)  Dynamic viscosity

\( \nu \)  Kinematic viscosity

\( \rho_{l} \)  Density of the base fluid (liquid steel)

\( \rho_{g} \)  Density of dispersed gas phase (argon gas)

\( \sigma \)  Electrical conductivity

\( \tau_{m} \)  Momentum response time of a particle

\( \chi \)  Ratio between the major and minor axes of a bubble

\( \Phi \)  Electric potential

\( \Delta \)  Size of the cell
Chapter 1

Introduction

1.1 Continuous Casting of Steel

Continuous Casting (CC) is used to produce more than 90% of steel in the world [1] in the form of semifinished shapes such as slabs, blooms, billets, beam blanks, and sheets. The basic process of CC is illustrated in Fig. 1.1(a). In this process, hot molten steel flows through a ladle, tundish, upper tundish nozzle (UTN), slide gate, submerged entry nozzle (SEN) into the mold region. The flow is driven by gravity and the flow rate is controlled by moving the eye-shaped restriction formed by horizontal translation of a slide gate, which is a simple ceramic valve consisting of a plate with a circular hole. The molten steel starts cooling in the mold region against a water-cooled copper mold and forms a solidified shell in the mold as shown in Fig. 1.1(b). The shell contains the molten steel as it is withdrawn from the bottom of the mold. Due to the large amount of steel produced, even small improvements to this process can have a big economical

Figure 1.1: Continuous casting process
(a) A schematic of the continuous casting process and (b) A close view of tundish, SEN and mold region of the caster [2]
impact. Most defects arise in the mold region of the casting process, due to the entrapment of inclusion particles into the solidifying shell, and crack formation in the newly-solidified steel shell. Thus, the first step to improve steel products is to fully understand the mechanisms of defect formation, and to find windows of safe operating conditions that avoid these problems. The harsh thermal environment makes experiments difficult, but computer models allow the process to be understood and improved through simulations.

1.2 Objectives of the research

The objective of the author's doctoral research is to utilize advanced computational techniques and parallel computers to model the turbulent flow in the continuous casting caster, and understand the fluid flow, transport and capture of inclusion particles and bubbles in the mold region under different casting conditions. During the continuous casting process, argon gas is commonly injected at the slide gate or stopper rod to avoid air aspiration, re-oxidation, and clogging problems. The jet of molten steel then carries the argon bubbles through the SEN and into the mold cavity. During this transport, the bubbles may pick up inclusion particles, thereby forming inclusion clusters which if captured in the solidifying shell would lower the steel quality [3, 4]. Therefore the dynamics of the argon bubbles and their interaction with small inclusion particles are important. At high casting speed, an Electromagnetic Braking (EMBr) system is often used in this process to alter the flow pattern and meniscus characteristics of fluid in the mold region. By employing coils with direct current, a static magnetic field is created. The magnetic field passes through the mold region and a current field is generated in the conducting steel. This induces a Lorentz force which modifies the flow in the mold cavity and the transport of argon bubbles and inclusion particles. This research consists the following objectives: develop a mathematical model for the molten steel flow and particle transport in the caster; develop efficient numerical tools to solve the mathematical model; use the developed models and numerical tools to study the effects of different casting parameters (casting speed, SEN submergence depth and EMBr) on the motion of molten steel and transport of argon bubbles; based on the modeling results, provide suggestions on operations to improve product quality.

1.3 Physical Phenomena and Literature Review

To model the transport and the capture of inclusions in the CC process, several important physical phenomena in this complex real world problem need to be considered. The first is the effect of the external magnetic field on the molten steel flow which significantly affects the flow pattern and the transport of the gas bubbles and inclusions. The second is the coupling between the argon gas and the molten steel. The
injection of buoyant argon gas modifies the flow pattern, especially under high argon injection rate. The third is the interaction of the particles/bubbles with the solidification front which decides the fate of the inclusions (whether captured or not) when they approach the solidification front. In this section, a brief review of the literatures on these topics is presented.

1.3.1 Effect of EMBr

The modeling of ruler-shaped EMBr effect on fluid flow in the CC process has been studied by many other researchers, including several at University of Illinois. \cite{5-7}. Chaudhary et al. \cite{5} studied the effect of single and double ruler EMBr and showed that applying a magnetic field can suppress turbulence. They found that with insulated walls (no steel shell), the flow is more unstable with EMBr compared to flow without the EMBr. Singh et al. \cite{6} later performed Large-Eddy-Simulation (LES) with the inclusion of a conducting shell for otherwise identical conditions of Chaudhary et al. \cite{5}. Results of Singh et al. \cite{6} showed that the large-scale jet wobble and the transient asymmetric flow in the mold with insulated walls as observed by Chaudhary et al. \cite{5} was not found with conducting walls. Singh et al. \cite{7} further investigated the effect of a double-ruler EMBr in a real caster. Their results again showed that the flow was more stable with the magnetic field and the surface velocities were smaller (\(\sim 0.1 \text{ m/s}\)). The surface level was flatter with EMBr and the level fluctuations were reduced (\(< \pm 1 \text{ mm}\)). The combined effect of local EMBr \cite{8} and SEN submergence depth on the motion of molten steel has been showed to be very important \cite{8}, however the combined effect of ruler-shape EMBr and SEN submergence depth has not been studied in detail.

1.3.2 Effect of Argon Gas Injection

Argon gas is commonly injected at the slide gate or stopper rod to avoid air aspiration and SEN clogging problems. Argon bubbles entering the mold region have three possible fates: (1) some reach the top surface, pass through the slag layer and escape harmlessly to the atmosphere; (2) some are captured by the solidifying shell near the meniscus and lead to surface defects owing to the many solid nonmetallic inclusion particles that are typically attached to each bubble; (3) some are captured deep in the caster and cause internal defects. In addition, the Ar bubbles may change the flow pattern through their interaction with the molten steel flow, and affect the trajectories of inclusions that are being transported in the caster. Some works have been reported on the study of Ar and molten steel flow in SEN and mold region of the caster \cite{4, 9-18}. Liu et al. \cite{9} used an Eulerian-Lagrangian two-way coupled (steel flow can affect the Ar gas flow and vice versa) transient 3D \(k - \epsilon\) model simulation on a grid with 150,000 cells and studied the effect of argon bubble transport in SEN and the mold region. Their results showed that inside the SEN the bubbles were close
to the wall, which could prevent clogging. They also found that in the mold region bubbles with diameters 0.25–2.5 mm move to top surface while bubbles smaller than 0.25 mm move to the narrow or wide surfaces. Pfeiler et al. [10] performed transient 3D $k - \epsilon$ Eulerian-Lagrangian simulations on a mesh with 500,000 cells to study the motion of Ar bubbles and non-metallic inclusions in 1/4 of a thin slab caster (135 mm thick). Through studies of one-way coupling (i.e. the argon bubbles do not affect the molten steel motion) and two-way coupling between the molten steel and argon bubbles, they observed that two-way coupled simulations are important for accurate prediction of bubble/particle trajectories, especially when there are larger bubbles. In addition to these studies using Lagrangian description for argon bubbles [4, 9–15], other researchers have applied Eulerian description for argon bubbles to investigate the two-phase flow in the caster [16–18]. By using the water model and Eulerian simulations, Singh et al. [16] showed that higher flow rates lead to larger and more bubbles. Their results also showed that the flow outside of the SEN would be asymmetric when argon flow rate was large. Li et al. [17] used a finite volume method and Eulerian description to study the Ar-steel flow. They showed that the argon injection leads to an increase of upward velocity in the region outside of SEN. Sanchez-Perez et al. [18] conducted water model experiments using Particle Image Velocimetry (PIV) and numerical simulations to study the effect of gas injection on the flow field in caster. They showed that the angle of both the liquid and the gas jet outside of SEN can be affected by the argon flow rate. In order to study the number, size and location of particles captured by the solidification front, the Eulerian-Lagrangian approach is more appropriate and is therefore considered in the proposed research. Yu and Zhu [19] did numerical simulations to study the multiphase phenomena in the presence of a magnetic field. They pointed out that argon gas injection can improve the removal rate of inclusion particles but EMBr does not help in floating up of small inclusion particles.

1.3.3 Inclusion Transport and Capture

Several researchers have investigated the capture rate and distribution of inclusions in the casting process [4, 12, 15]. Yuan et al. [12] performed one-way coupled particle capture simulations for small bubbles (<40 $\mu$m) in a thin-slab steel caster. Large Eddy Simulations were used to simulate the molten steel flow and the effect of turbulence on particle transport were automatically included. Their study also focused on particles with diameter less than the PDAS and therefore the “simple” criterion was implemented in their simulation. Their results predicted a removal rate of 8% of small inclusions, and they found that the removal fraction was independent of both particle size and density which indicated that the injected particles in their study were too small to deviate significantly from the surrounding fluid flow.
Zhang and Wang [15] performed 3D one-way coupled $k-\epsilon$ simulations with a sink term approach (sink terms were added at the solidification front to include the effect of mass and momentum loss during solidification [12]) and full solidification approach (solve for the liquid fraction to identify the mushy zone) to predict the particle entrapment in a full length of a billet caster. To include the effect of dispersion of particles due to turbulence, a random walk model [20] was applied. They studied the transport and capture of small (5 µm) particles. In the sink term approach, they assumed an immediate capture when particles touched the pre-defined shell. This is reasonable when the particle diameter is smaller than the PDAS [4, 12–14]. In the full solidification approach, they assumed an immediate capture when particles reached a cell with liquid fraction between an empirical range (0.3 to 0.6). Their model predicts less inclusions captured at the center of the billet than the surrounding region which disagreed with experiments. They thought the inaccuracy of their model was caused by ignoring the effect of dendrite arm spacing and only 5 µm inclusions were considered in their simulation.

A recent work done by Thomas et al. [4] applied a force balance capture criterion to investigate the entrapment of slag inclusions (40, 100 and 400 µm), alumina cluster (100 µm) and bubbles (100 and 2500 µm) during the continuous casting of steel slabs. Their results showed that the overall particle removal rate was less than 20% and the gas injection helps to increase particle removal rate at the top surface for small bubbles around 100 µm. They also found that to more accurately predict the removal rate of particles (error within ±3%), more than 2500 particles should be injected into the domain.

The above literature review shows that the effect of different values of EMBr and combination with SEN submergence depth on the molten steel flow is currently not well understood and the optimal setup of those two parameters is not clear. Only a small number of previous works have studied the inclusion transport in the caster, and even less work has been done using high-fidelity two-way coupled techniques which include the effect of argon gas on the fluid flow. The effect of EMBr and SEN submergence depth on particle capture in a real caster has also not been studied in detail.

1.3.4 Bubble and Particle Capture at Solidification Front

In order to accurately predict the capture rate and location of Ar bubbles/particles, a suitable capture criterion needs to be applied. For small inclusions, a “simple” capture criterion which assumes immediate capture of inclusions when they touch the solidification front has been used by some researchers.[12, 15] Several previous works have investigated the interaction of a particle in front of a planar solidification front [21–27]. Many of these works [21–23] are fundamental studies with simplified conditions (i.e. planar solidification front, stationary flow, etc). Rempel and Worster [21] derived an analytical expression for the velocity of a
particle in front of a propagating solidification front, and derived a criterion that if the solidification velocity is greater than a maximum particle velocity then the particle will be captured. A multiscale model was proposed by Garvin et al. [22] to simulate the solidification front and particle interactions when the particle is in micron size range. To investigate the interaction of a spherical particle and an advancing solidification front, Kao et al. [23] applied numerical boundary integral and continuation methods to determine the critical speed for particle capture, and their results showed that for binary solidification the particle speed is one order of magnitude below those for a single component system. However, these [21–23] proposed capture criteria are not suitable for direct application in continuous casting because (1) the fluid flow in the caster is strongly inertia driven with large cross flow near solidification front; and (2) the solidification front of steel in the caster is of dendritic shape which is more complex than a planar shape, so the interaction is more complex. For the case of a particle interacting with a dendritic solidification front, three possible interaction modes have been proposed: pushing, entrapment and engulfment [24].

There are several existing experimental works on particle [24] and bubble [25] interactions with a dendritic solidification front. Wilde and Perepezko [24] performed several experiments for solidifying liquid of different combinations of Cu and Ni with dispersed nonmetallic particles. Their results showed that the critical velocities for particle engulfment are more than three orders of magnitude larger than those predicted by kinetic models. This indicates the dendritic solidification front can lead to a preferred pushing of larger particles. Xing et al. [25] conducted solidification experiments using succinonitrile based transparent alloys (SCN-1.5 wt%ACE) to investigate bubble engulfment and entrapment by cellular and dendritic interfaces during directional solidification. They observed weak solid-gas coupled growth in their experiments when the solidification velocity was high.

Yuan [13] and Thomas et al. [4] developed a local force balance based particle capture criterion for predicting bubble/particle capture by the dendritic solidification front in the continuous casting process. Their proposed capture criterion considered the effects of particle properties including Primary Dendrite Arm Spacing (PDAS), local flow field, local concentration gradients, surface tension effects, and other forces [4]. In this criterion, particles with diameter smaller than the PDAS are captured immediately when the touch the solidification front. Comparing with other kinetic models, this capture criterion is more suitable for studying the capture of Ar bubbles and inclusions in the continuous casting process.

1.4 References


Chapter 2
Mathematic Models and CUFLOW

As presented in previous chapter, using RANS $k-\epsilon$ model for the turbulent liquid steel flow and Lagrangian representation for argon bubbles can be used to study the transport and capture of the argon bubbles and inclusion particles. [1–4] However, to include the effect of turbulence dispersion on bubbles and particles, this approach relies on the use of a stochastic model such as the random walk model. [1–4] The random walk model assumes isotropic turbulence and has been showed to largely over-predict particle deposition in aerosol flows. [5] The model can be improved by adding near-wall corrections [5], however it still over-predict the deposition of small particles and the near wall correction is very sensitive to the choice of the cut-off distance (how far away from the wall to use the correction). [6]. Further more, the flow field in the caster is usually transient and both high and low frequency oscillations has been observed in previous research, especially in cases with a wide mold. [7–10] The RANS approach cannot capture these transient phenomena, thus hardly to accurately predict the transport and capture of the argon bubbles and inclusions. Therefore, a more accurate but computational extensive large eddy simulation (LES) model is used. Part of the author’s contribution is implemented the LES module, magnetohydrodynamics module, and two-way coupled Eulerian-Lagrangian module into CUFLOW. Solving the transient flow with LES approach on a fine grid and simultaneously tracking the trajectory of millions of Lagrangian particles is computational intensive and requires more memory than a single GPU can support. Therefore, the author further developed a multi-GPU module which utilizes the power of several GPUs to accelerate the computation. Section 2.1 to 2.5 introduce the mathematical models, and Section 2.6 discusses the multi-GPU implementation of CUFLOW and its speedup.

2.1 Navier-Stokes Equations and Fraction-Step Method

The motion of incompressible Newtonian fluid flow is governed by the Navier-Stokes equations:

$$\nabla \cdot (\rho u) = 0$$

(2.1)
\[ \frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot \left[ \mu (\nabla \mathbf{u} + \nabla \mathbf{u}^T) \right] + \mathbf{S} \] (2.2)

where \( \mathbf{u} \) is fluid’s velocity, \( t \) is time, \( \rho \) is fluid’s density, \( \mu \) is fluid’s dynamic viscosity, \( p \) is total pressure, \( \mathbf{S} \) is other source terms, \( \mathbf{u} \otimes \mathbf{u} \) is such that \( \mathbf{u} \times \mathbf{u}^T \). By combing the convection and diffusion terms into a single term represented by \( \mathbf{H} \), we have:

\[ \frac{\partial \rho \mathbf{u}}{\partial t} = -\nabla p + \mathbf{H} + \mathbf{S}. \] (2.3)

In this work, the above equations are solved using a fractional-step projection method. The fractional-step projection method was initially introduced in 1967 by Chorin [11, 12] and Temam [13, 14]. Since than, it has been widely used to solve the three dimensional time dependent Navier-Stokes equations.

In the first step of the fractional step method, intermediate velocities are determined by solving the momentum equations without the pressure gradient terms. The discretized equations are derived by a finite-volume framework using central differencing for both convection and diffusion terms on a collocated grid. For the temporal differencing, the second-order accurate Adams-Bashforth scheme is used.

\[
\left( \frac{\rho u_i^{n+1} - \rho u_i^n}{\Delta t} \right) = -\nabla p_i^{n+1} + \frac{3}{2} H_i^n - \frac{1}{2} H_i^{n-1} + S_i^n
\] (2.4)

After a time-splitting on Equation 2.4, an expression for the intermediate velocity, \( \hat{u}_i \), is obtained in the absence of pressure:

\[
\left( \frac{\rho \hat{u}_i - \rho u_i^n}{\Delta t} \right) = \frac{3}{2} H_i^n - \frac{1}{2} H_i^{n-1} + S_i
\] (2.5)

and the velocity for the next time step \( u_i^{n+1} \) can be obtained by:

\[
\left( \frac{\rho u_i^{n+1} - \rho \hat{u}_i}{\Delta t} \right) = -\nabla p_i^{n+1}
\] (2.6)

Before this step, the continuity equation is transformed to a pressure-Poisson equation given by:

\[
\frac{\rho}{\Delta t} \frac{\partial \hat{u}_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\partial p}{\partial x_i} \right)^{n+1}
\] (2.7)

Equation (2.7) for pressure can be solved efficiently by a V-cycle multigrid method, and red-black SOR (with over-relaxation parameter of 1.6). After computing the pressure at \( n + 1 \) time step from equation (2.7), the velocity components are updated for the effects of the pressure gradient term. For steady state calculations,
the algorithm is marched in time to desired degree of convergence in time.

### 2.2 Large Eddy Simulation Model

Using Direct Numerical Simulation (DNS), the above continuity and momentum equations can be numerically solved without any turbulence model. However, this requires to resolve the whole range of temporal and spatial scales of the turbulence. All the spatial scales of the turbulence must be resolved in the computational mesh, from the dissipative scales (Kolmogorov microscales) to the integral scale (associated with the motions containing most of the kinetic energy). Thus, DNS is computationally very expensive, especially for high Reynolds numbers. Therefore, turbulence models are used in this work to reduce the computation cost.

One turbulence model used in this work is the Large Eddy Simulation (LES) model. In the LES approach, the sub-grid scale eddy viscosity $\mu_{sgs}$ is used to model the influence of the turbulent scales that are not resolved explicitly. After introducing the sub-grid scale viscosity, the momentum equation (6.2) can be rewritten as:

$$\frac{\partial \rho u}{\partial t} + \nabla \cdot (\rho u \otimes u) = -\nabla p + \nabla \cdot \left[ (\mu + \mu_{sgs}) (\nabla u + \nabla u^T) \right] + S \quad (2.8)$$

and $\mu_{sgs}$ is modeled by the Coherent-Structure Smagorinsky Model (CSM) [15]. The $\otimes$ operation means a tensor product (or outer product). The CSM model does not need a wall-damping function. In CSM, first the Smagorinsky eddy-viscosity model is used to compute $\mu_{sgs}$:

$$\mu_{sgs} = \rho \nu_{sgs} = \rho C_s^2 (\Delta x \Delta y \Delta z)^{2/3} \sqrt{S_{ij} S_{ij}} \quad (2.9)$$

where $\Delta x$, $\Delta y$ and $\Delta z$ are the cell dimensions in $x$, $y$ and $z$ directions, respectively. $C_s$ is the Smagorinsky constant which is computed as

$$C_s^2 = C_{cs} |Q/E|^{3/2} (1 - Q/E) \quad (2.10)$$

where $C_{cs}$ is a model constant equal to 1/22. $Q$ is the second invariant as given by Eqn. (2.11), and $E$ is given by Eqn. (2.12).

$$Q = 1/2 (W_{ij} W_{ij} - S_{ij} S_{ij}) \quad (2.11)$$

$$E = 1/2 (W_{ij} W_{ij} + S_{ij} S_{ij}) \quad (2.12)$$

$S_{ij}$ and $W_{ij}$ are the filtered velocity-strain tensor and vorticity tensor, respectively. The CSM model damps
the eddy viscosity close to the wall and also automatically incorporates the effect of anisotropy induced by the applied magnetic fields on the filtered sub-grid scales.\cite{16} Therefore, no additional modifications to account for anisotropic subgrid effects are added. This model has been successfully tested and used previously in predicting fluid flow in steel casters with magnetic fields. \cite{17, 18}

\subsection{2.3 Magnetohydrodynamics Model}

In the continuous casting process, an Electromagnetic Breaking (EMBr) system is often used in this process to alter the flow pattern and meniscus characteristics of fluid in the mold region. By employing coils with direct current, a static magnetic field is created. The magnetic field passes through the mold region and a current field is generated in the conducting steel. This induces a Lorentz force that acts on the molten steel and modifies the flow in the mold cavity.

Depending on the strength of the coupling between the applied and induced magnetic fields, there are two methods to model the electromagnetic effects, namely “electrical potential method” and “magnetic induction method”. The electric potential method assumes that the induced magnetic field is much smaller than the external magnetic field and therefore the former can be ignored. Based on this assumption, the current can be obtained from solving an equation for the electric potential. This approach is applicable for a static magnetic field and lower magnetic Reynolds numbers. The magnetic Reynolds number is defined as

\[ Re_m = \frac{\mu_e \sigma L u}{\mu_e} \]

where \( \mu_e \) is the magnetic permeability of free space, \( \sigma \) is the conductivity of the liquid steel, \( L \) is the typical length scale and \( u \) is velocity scale. However, when a large \( Re_m \) or time varying magnetic field is considered in the problem, the magnetic induction method should be used and the induced magnetic field need to be solved as well \cite{19}. In the continuous casting process with EMBr, \( Re_m \) is usually small (\( Re_m \approx 0.1 \)), and the electric potential method is widely used \cite{17, 20, 21}.

The Lorentz force \( \mathbf{F}_L \) is obtained by taking cross product of current density \( \mathbf{J} \) and external magnetic field \( \mathbf{B} \), as shown in Eqn. (2.13). The electrical current density \( \mathbf{J} \) can be computed through the Ohm’s law as given by Eqn. (2.14), and for a well conducting material the current conservation law is given by Eqn. (2.15). Therefore, the electric potential \( \Phi \) satisfies Eqn. (2.16). It is to be noted that the electrical conductivity of gas and liquid have different values (the conductivity of liquid steel is \( \sim 10^{20} \) larger than that of argon gas) and therefore the electrical conductivity \( \sigma \) in Eqn. (2.16) cannot be canceled. With a insulated wall, the boundary condition for \( \Phi \) is \( \partial \Phi / \partial n = 0 \).

\[ \mathbf{F}_L = \mathbf{J} \times \mathbf{B} \quad (2.13) \]
\[ J = \sigma ( -\nabla \Phi + u \times B ) \quad (2.14) \]

\[ \nabla \cdot J = 0 \quad (2.15) \]

\[ \nabla \cdot (\sigma \nabla \Phi) = \nabla \cdot [\sigma (u \times B)] \quad (2.16) \]

### 2.4 Two-way Coupled Eulerian-Lagrangian Particle Tracking

In this work, a two-way coupled Eulerian-Lagrangian model to include the effects of discrete phases (argon bubbles and inclusion particles). This approach takes into account both the effect of the argon bubbles on molten steel flow and the effect of steel flow on argon bubble transport. One advantage of using Eulerian-Lagrangian method is that instead of using only the averaged bubble size (such as in Eulerian-Eulerian approach), bubbles of different sizes can be injected simultaneously which is more realistic of what happens in the real casting process. The other advantage is that such an approach enables the study of the distribution of captured bubbles of different sizes, since all the argon bubbles are tracked as Lagrangian particles.

#### 2.4.1 Particle Motion Equations and Forces on Particle

To study the capture locations and the distribution of bubbles/particles of different sizes, a Lagrangian description of the inclusions will be used. The argon bubbles are modeled as discrete Lagrangian points that are carried by the molten steel. Collision between these bubbles and particles, as well as breakup and coalescence of the argon bubbles are ignored. Since the volume fraction of the argon gas is low (usually less than 8%), this method is applicable. The motion of the individual gas bubbles and particles is governed by Newton’s second law given by Eqn. (2.17)

\[ m_p \frac{du_p}{dt} = \sum F_{pT} \quad (2.17) \]

\[ \frac{dx_p}{dt} = u_p \quad (2.18) \]

where, \( m_p \) denotes the particle/bubble mass, \( u_p \) denotes particle/bubble velocity vector, \( x_p \) denotes the particle/bubble location, and \( F_{pT} \) denotes the total force vector exerted on the bubble/particle. In this
work, five forces are considered when the individual bubble/particle is transported in the bulk fluid. These forces are drag force $F_{pD}$, lift force $F_{pL}$, pressure gradient force $F_{pP}$, virtual mass force $F_{pV}$, and gravitational and buoyancy force $F_{pB}$. The implementations of these forces are presented below.

The drag force, $F_{pD}$, is the most important force and it represents the “steady-state” (the relative velocity of the particle to the base fluid is fixed) drag force that acts on the particle/bubble in a uniform pressure field. It can be written as shown in Eqn. (2.19), where $Re_p$ is the particle Reynolds number defined as $Re_p = \rho d_p \mu^{-1} |u - u_p|$ and $V_p$ is the volume of the particle.

$$F_{pD} = \frac{C_D}{24} \frac{18\mu}{d_p^2} V_p (u - u_p) Re_p$$

(2.19)

The drag coefficient of the argon bubble is modeled following Kuo and Wallis [22] and is given by Eqn. (2.20).

$$C_D = \begin{cases} 16Re_p^{-1} & Re_p \leq 0.49 \\ 20.68Re_p^{-0.643} & 0.49 \leq Re_p \leq 100 \\ 6.3Re_p^{-0.385} & 100 \leq Re_p \\ We/3 & 100 \leq Re_p \text{ and } Re_p < 2065.1We^{-2.6} \\ 8/3 & 100 \leq Re_p \text{ and } We < 8 \end{cases}$$

(2.20)

where $We = \rho d_p \gamma^{-1} |u - u_p|^2$, $\gamma$ denotes the surface tension between the argon gas and liquid steel. The advantages of this drag law is that it includes the shape of the bubble into the drag coefficient implicitly.

The lift force, $F_{pL}$, is caused by the rotation of the bubble/particle. When a velocity gradient exists around the particle, the particle may rotate and generate a lift force. The lift force can be expressed as shown in Eqn. (2.21) following Legendre and Magnaudet [23]. $C_L$ denotes the lift coefficient and is given by Eqn. (2.22).

$$F_{pL} = C_L \rho V_p (u_p - u) \times (\nabla \times u)$$

(2.21)

$$C_L = \sqrt{(C_L^{lowRe})^2 + (C_L^{highRe})^2}$$

(2.22)

where, $C_L^{lowRe}$ and $C_L^{highRe}$ are computed following Eqn. (2.23)

$$C_L^{highRe} = \left( \frac{1}{2} \right) \frac{1 + 16Re_p^{-1}}{1 + 29Re_p^{-1}} \quad \text{and} \quad C_L^{lowRe} = 6\pi^{-2} (SrRe_p)^{-0.5} J'(\epsilon)$$

(2.23)

where $J'(\epsilon) = 2.55 (1 + 0.2\epsilon^{-2})^{-3/2}$, $\epsilon = \sqrt{SrRe_p^{-1}}$, and $Sr = d_p |\nabla \times u| / |u - u_p|$ is the ratio between
the velocity difference across the bubble and the relative velocity. The lift force is more important when the density of the fluid is higher than the inclusion particle (in Eqn. (2.21), RHS $\rho V_p$ can be written as $\frac{\rho}{\rho_p} m_p$).

The virtual mass force, $F_{pv}$, also known as added mass force or apparent mass force, is related to the force required to accelerate or decelerate the liquid surrounding the particle/bubble. It is given by Eqn. (2.24) following Auton et al. [24] and Crowe et al. [25]. $C_V$ is the virtual mass coefficient and is unity according to Michaelides and Roig [26].

\[
F_{pv} = 0.5 C_V \rho V_p \left( \frac{Du}{Dt} - \frac{du_p}{dt} \right) \tag{2.24}
\]

The pressure gradient force, $F_{pp}$, is caused by the nonuniform pressure distribution around the particle/bubble, and is computed using Eqn. (2.25). This force is important if there is a large pressure gradient in the flow and if the particle density is smaller than or similar to the fluid density.

\[
F_{pp} = \rho V_p \frac{Du}{Dt} \tag{2.25}
\]

The buoyancy force is given by Eqn. (2.26), where $g$ is acceleration due to gravity.

\[
F_{pb} = g V_p (\rho_p - \rho) \tag{2.26}
\]

The first four of these forces comprise the source term $F_p$ in the momentum equation (Eqn. (6.2)). $F_p$ represents a source term exerted on the fluid from the dispersed phase and it has units of force per unit volume. The relation between $F_p$ and $F_{pT}$ is given by Eqn. (2.27).

\[
F_p = -V_{cell}^{-1} \sum_{i=1}^{n} (F_{pT,i} - F_{pb,i}) = -V_{cell}^{-1} \sum_{i=1}^{n} (F_{pD,i} + F_{pL,i} + F_{pv,i} + F_{pp,i}) \tag{2.27}
\]

where, $V_{cell}$ is the volume of the cell and $n$ is the number of particle or bubble in the cell.

### 2.4.2 Effects of Ferro-static Pressure on Bubble Size

When argon bubbles travel inside the cater, their sizes are affected by the ferrostatic pressure. This subsection presents a simple model to include the effects of pressure on bubble size variation. In this model, the argon gas is treated as a hypothetical ideal gas, where the ideal gas law is used as its equation of state:

\[
P V = nRT \tag{2.28}
\]
where $P$ is the pressure of the gas, $V$ is the volume of the gas, $n$ is the amount of substance of gas (in moles), $R$ is the universal gas constant, equal to the product of the Boltzmann constant and the Avogadro constant, and $T$ is the temperature of the gas.

Assuming nearly constant temperature, and by knowing the initial bubble radius $r_0$, the mass of the bubble $m_p$ and pressure $p_0$ placed $z_0$ below the liquid steel and slag interface, we compute the diameter $d_p(z)$ and density $\rho_p(z)$ of the bubble when it reaches to a new location which is $z$ below the top surface by:

$$d_p(z) = 2r_p(z) = 2\left[\frac{p_0}{\rho_l g (z - z_0) + 1.01 \times 10^5}\right]^{1/3} r_0 \tag{2.29}$$

$$\rho_p(z) = \frac{3m_p \rho_l g (z - z_0) + 1.01 \times 10^5}{4\pi p_0 r_0^3} \tag{2.30}$$

where $\rho_l$ is the steel density. This model is implemented into FLUENT using user defined functions. A testing case of is carried out to compute a 1 mm argon bubble rise in liquid steel from 1.5 m below the meniscus, where a atmosphere pressure is 101 kPa. The predicted bubble diameter and density are shown in Figure 2.1 The black dot lines in Figure 2.1 are computed using equations (2.29) and (2.30), while the solid red lines are output from FLUENT. Note the pink region shows the region above the port, and most large bubbles travel inside this region. For a 1 mm argon bubble, the diameter change is less than 5% in the region above the port. Note that for small bubbles, the surface tension effect is much more important, thus the effect of pressure on bubble diameter is even smaller.

### 2.4.3 Breakup and Coalescence of Argon Bubbles

As shown in Figure 2.2, gas pockets often form below the slide gate and inside the ports. [27, 28] A larger gas pocket is often seen to form in the low pressure back-flow region right below the slide gate, due to argon

![Figure 2.1: (a) Bubble diameter as a function of $z$ (b) Bubble density as a function of $z$](image-url)
bubbles accumulation. The strong shear flow continuously shears off small bubbles from the gas pocket and leads to different size distribution of argon bubbles. These bubbles keep moving downward and enter the ports where they interact with the swirling flow. In some cases, gas pockets form at the top of the ports due to back flow inside the ports. The mechanisms of bubble-bubble coalescence and breakup can be summarized in five categories: (a) coalescence due to collisions driven by the turbulence flow; (b) coalescence due to wake entrainment; (c) breakup due to interaction with turbulent eddies; (d) breakup of large cap bubbles due to interface instability; (e) shearing off of small bubbles from large cap/slug bubbles. In the continuous casting process, the argon fraction is small (<10%) therefore no cap or slug bubbles exist. However, the shearing of small bubbles from the gas pockets may happen. The surface tension between argon and steel is $\sim 1.2$ N/m which is $16 \times$ larger than that of air water system. This also reduces the possibility of breakup due to interaction with turbulent eddies, and coalescence of argon bubbles due to collision. The possibility of the coalescence wake entrainment is also low because the bubbles in the caster are usually small ($\sim 1$ mm). However, the coalescence of argon bubbles may happen at some regions with very high local argon volume fraction (e.g. below the slide gate). This work focus on the transport of small argon bubbles inside the caster where the argon volume fraction is low. Therefore, breakup and coalescence of bubbles are ignored in this work.

### 2.5 Volume of Fluid (VOF) Model

In the Eulerian-Lagrangian approach, the drag force on the discrete phase is computed using the drag coefficient as previously shown in Eqn. (2.20). However, when argon bubble moves in the liquid steel with magnetic file applied, the drag coefficient on the bubble is different and it is known that bubble moves slower in this case and therefore a larger drag coefficient is expected. In most cases, drag coefficient can be obtained through experimental measurements or computational model. Since well conducting liquids are

![Figure 2.2: Gas pockets in below the slide gate (left) and at the top of port (right)](image-url)
usually metals and thus opaque, experiments to measure the bubble velocity have been difficult. Therefore, VOF approaches, which can resolve the shape and motion of the bubble, are used to find out an appropriate drag coefficient that can be used to compute the drag force on a particle that moving in a conducting liquid under the presence of a horizontal magnetic field.

To mitigate spurious velocities normally generated during numerical simulation of multiphase flows with large density differences, an improved algorithm for surface tension modeling, originally proposed by Wang and Tong [30] was implemented by Kumar and Vanka [31]. In this approach, the surface tension force \( \mathbf{F}_S \) (which goes to the right hand side of the momentum equation), is evaluated using Eqn. (2.31).

\[
\mathbf{F}_S = \int_{\Gamma} \gamma \kappa \mathbf{n} \delta (\mathbf{x} - \mathbf{x}_f) \, ds
\]  

(2.31)

where \( \gamma \) and \( \kappa \) denote the surface tension and mean interface curvature, respectively. \( \Gamma \) represents the interface, \( \mathbf{n} \) denotes the normal vector of the interface, \( \delta \) is the Dirac delta function. \( \mathbf{x} \) and \( \mathbf{x}_f \) denote the coordinates of the cell and the interface, respectively. To capture a sharp interface and reduce spurious velocities, a Sharp Surface Force (SSF) method for modeling of the surface tension force is adapted. This SSF method, also known as Pressure Boundary Method (PBM) or Ghost Fluid Method (GFM), has been presented and discussed in detail elsewhere [30–33]. In this method, the surface tension in Eqn. (2.31) is treated as a pressure gradient \( -\nabla \tilde{p} \) which exactly balances the surface tension force \( \mathbf{F}_S \) generated due to presence of the interface. By considering that this new pressure field cannot generate velocity in a static case, another pressure Poisson equation can be obtained as given by Eqn. (2.32) below.

\[
\nabla \cdot \left( \frac{\nabla \tilde{p}}{\rho} \right) = F_x^+ - F_x^- + F_y^+ - F_y^- + F_z^+ - F_z^-
\]  

(2.32)

where the six terms on the RHS are scalars and defined as

\[
F_x^+ = -\left[ \frac{\gamma \kappa}{\rho \Delta x^2} \right]_{(i+1/2,j,k)}
\]  

(2.33)

\[
F_x^- = -\left[ \frac{\gamma \kappa}{\rho \Delta x^2} \right]_{(i-1/2,j,k)}
\]  

(2.34)

and etc. The surface tension force at the interface is thus treated as a jump condition for calculating this new pressure field. The interface is tracked using the VOF method in which an evolution equation for the
liquid volume fraction $\alpha$, given by Eqn. (2.35) is solved.

$$\frac{\partial \alpha}{\partial t} + u \cdot \nabla \alpha = 0$$  \hspace{1cm} (2.35)

A physical property $\theta$ (i.e. $\rho$, $\mu$ and $\sigma$) at a given point in the domain is evaluated by linear interpolation as $\theta = \alpha \theta_l + (1 - \alpha) \theta_g$, where the subscript “l” denotes the property of surrounding liquid and subscript “g” denotes the property of the gas phase.

### 2.6 In-house Code CUFLOW and GPU Computing

All the above models are implemented into a in-house code CUFLOW. The implementation of the LES model, two-way coupled Eulerian-Lagrangian model and MHD model belongs to part of this work. CUFLOW [34–37] is a general purpose code for simulating laminar and turbulent flows in complex domains. The code employs Cartesian grids to integrate the three-dimensional unsteady incompressible Navier-Stokes equations. The continuity and momentum equations are solved using a fractional step method. Previous version of CUFLOW utilized a staggered data storage. The new version of CUFLOW employs a collocated grid data structure, in which both the pressure and velocity field are stored at the cell centers. CUFLOW uses Graphics Processing Units (GPU) to solve the discretized equations in parallel.

Three Poisson type equations are involved in the solution procedure. Both direct (e.g. Gauss elimination and LU decomposition) and iterative methods (e.g. Jacobi, Gauss-Seidel and conjugate gradient) can be used to solve these elliptic type equations. For a very large three dimensional system, direct methods are rarely used due to its extremely high computation cost. In this work, a multi-grid red-black successive over relaxation (SOR) method was implemented using CUDA Fortran [38] onto GPUs to solve the Poisson equations. This section first introduces the architecture of a GPU, then shows an example of GPU implementation of red-black SOR. Finally, discusses the implementation on multiple GPUs.

#### 2.6.1 GPU and Its Architecture

What is a GPU? GPU is a graphics processing unit that was initially designed to manipulate and alter memory to accelerate the creation of images in a frame buffer intended for output to a display. It is widely used in computer, mobile devices and game consoles. As the name implies, they were primarily used in graphic processing such as video gaming, audio and video coding. Driven by the insatiable market demand for real-time, ultra high-definition 3D graphics editing and 3D gaming, the GPU has evolved into a highly parallel, multi-threaded, manycore processor with tremendous computational horsepower and very high memory.
bandwidth. GPUs become popular in the field of scientific computing after Nvidia introduced the Compute Unified Device Architecture (CUDA), a new parallel computing platform and programming model, in 2007. Lots of supercomputers (e.g. NCSA BlueWaters, ORNL Titan and Chinese Tianhe-2) are now equipped with thousands of GPUs as co-processors. Figure 2.3 shows a Nvidia K40 GPU, which has a dedicated memory of 12 GB and memory bandwidth of 288 GB/s. It has a theoretical peak computation power of 4.29 TFLOPS in single precision and 1.43 TFLOPS in double precision.

Figure 2.3: A Tesla K40 GPU made by Nvidia

GPU architecture is quite different from that of a CPU. The Tesla K40 GPU uses the Kepler GK110 compute architecture, and it has 7.1 billion transistors. The basic computational unit on the GPU is a thread processor, also referred to as a “core”, and one thread processor is a floating-point unit. In the GPU, many thread processors are grouped into streaming multiprocessors (SMX), which contain a limited amount of resources used by resident threads, namely registers and shared memory. Threads are also being grouped into warps, and each warp contains 32 threads. Warp is the actual grouping of threads that gets calculated in single-instruction, multiple-thread (SIMT) fashion. Each instruction on the device is issued to a warp of threads, and execution of instructions is performed by each thread in a warp in lockstep. Different warps in a thread block may be executing different instructions of the device code, and all of this activity is coordinated behind the scenes by the scheduler on each SMX. A full Kepler GK110 implementation includes 15 SMX units and six 64bit memory controllers. Figure 2.4 (from Nvidia GK110 Data Sheet) shows the chip block diagram of a single SMX. Each SMX consists of 192 single precision cores and 64 double precision unit, which results in 2880 cores and 96 double precision unit in the total 15 SMX units.

2.6.2 Programing on GPU and GPU Speedup

The old version of CUFLOW uses CUDA C as the programing language, while this new version utilize the CUDA Fortran [38]. Same as CUDA C, CUDA Fortran is also a hybrid programming model, meaning that code sections can execute either on the CPU or the GPU. The terms host is used to refer to the CPU and
its memory, and the term device is used to refer to GPU and its memory. A subroutine that executes on the device but is called from the host is called a “kernel”, and the kernel is marked with CUDA keywords for labeling data-parallel functions.

The programming procedure are illustrated using an example: solving a steady state heat conduction problem on a three dimensional domain (simply as a cube with Dirichlet boundary condition) with red-black SOR. The dimension less governing equation for a steady state 3D heat conduction is simply a Laplace equation:

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} = 0$$ (2.36)

where $T$ is the temperature. Discretize the equation using finite difference method with a center differencing scheme, and apply SOR we get:

$$T_{i,j,k} = \omega \times \frac{\varepsilon_{i,j,k}}{C_{i,j,k}} + (1 - \omega) \times T_{i,j,k}$$

$$\varepsilon_{i,j,k} = C_{i-1,j,k}T_{i-1,j,k} + C_{i+1,j,k}T_{i+1,j,k} + C_{i,j-1,k}T_{i,j-1,k} + C_{i,j+1,k}T_{i,j+1,k} + C_{i,j,k-1}T_{i,j,k-1} + C_{i,j,k+1}T_{i,j,k+1}$$ (2.37)

$T_{i,j,k}$ is temperature, $C_{i,j,k}$ coefficient, $\omega$ is relaxation factor. A converged solution is achieved when the
following condition is satisfied:

\[ \Omega = \sum_{(0,0,0)}^{(n_x,n_y,n_z)} \left[ \varepsilon_{i,j,k} - C_{i,j,k}T_{i,j,k} \right] < \Omega_0 \]  \tag{2.38}

where \( \Omega_0 \) is the maximum global residual allowed. Examine equation (2.37) shows that this process can be parallelized by using a red-black labeling shown in Figure 2.5. The grid points are colored with black and red colors in three dimensional space. In 2D it looks like a checkboard. With this coloring, the points with same color can be updated concurrently (no race condition). The procedure of solving equation 2.37 is shown in Figure 2.6. Note that, the computation of global residual \( \Omega \) requires an atomic operation. Here, in this simple example, the summation is done on the host CPU. Several tests are carried out to test the performance of red-black SOR on different CPUs and GPUs. To fairly compare the performance, the algorithm is implemented using ANSI C code and parallelized using both MPI and OpenMP. In the testing case, a large grid consists of 256 grid points on each direction (total 16 million points) is used to discretize the Laplace equation. The speedup and the price of the CPU and GPU are listed in Table 2.1. The results shows the maximum speedup obtained is using the Tesla K40 which provides is 9.9 times faster than the
Table 2.1: Speed up using different CPU and GPU with problem size: $N_x \times N_y \times N_z = 256 \times 256 \times 256$

<table>
<thead>
<tr>
<th>GPU Name or CPU Name</th>
<th>Time for Data Copy to/from Device (s)</th>
<th>Time for 1000 iterations (s)</th>
<th>Speed up</th>
<th>Price* ($)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Xeon X5650</td>
<td>-</td>
<td>386 (1 thread)</td>
<td>1</td>
<td>500</td>
</tr>
<tr>
<td>Xeon X5650</td>
<td>-</td>
<td>129 (MPI 2x2x2)</td>
<td>3</td>
<td>500</td>
</tr>
<tr>
<td>Xeon E5 2670v2</td>
<td>-</td>
<td>117 (MPI 2x2x2)</td>
<td>3.3</td>
<td>1500</td>
</tr>
<tr>
<td>Xeon E5 2670v2</td>
<td>-</td>
<td>98 (OMP 10 Threads)</td>
<td>4</td>
<td>1500</td>
</tr>
<tr>
<td>Xeon E5 2670v2</td>
<td>-</td>
<td>127 (OMP 20 Threads)</td>
<td>3</td>
<td>1500</td>
</tr>
<tr>
<td>Tesla K40</td>
<td>0.26 / 0.04</td>
<td>39</td>
<td>9.9</td>
<td>3300</td>
</tr>
<tr>
<td>Tesla C2075</td>
<td>0.59 / 0.11</td>
<td>88</td>
<td>4.4</td>
<td>1000</td>
</tr>
<tr>
<td>GTX 660M</td>
<td>0.39 / 0.07</td>
<td>220</td>
<td>1.8</td>
<td>150</td>
</tr>
</tbody>
</table>

*Price data come from Newegg, Rakuten and Ebay, Dec. 6, 2015.

The time for sequential execution on an Intel Xeon X5650 CPU. Using 10 OpenMP threads on a Xeon E52670v2 CPU gives a maximum speedup of 4, which is half of the performance produced by Tesla K40. Note the Xeon E52670v2 CPU price is also about half of that of Tesla K40.

To check the speedup dependence on problem size, another six testing problem are performed on Nvidia Tesla K40 and the Intel Xeon E52670v2. The speed up is shown in Table 2.2 In the cases with small problem size, the sequential execution on CPU is the fastest. Using 10 OpenMP threads is slower than the sequential execution due to the overhead in OpenMP. Similarly, due to the data copy and kernel overhead on the Tesla GPU, running on GPU is slower than sequential execution on the CPU. However, with the increase of problem size the GPU is running more efficient and eventually becomes $7.3 \times$ faster than sequential execution and $2.5 \times$ than the best performance achieved on the single CPU. Those speedup results presented in Table 2.1 and Table 2.2 are also illustrated in Figure 2.7. The results shows when the problem size is greater than $32 \times 32 \times 32$, the speedup of using GPU is greater than 1.0. Therefore, GPU provides higher parallel efficiency with a larger problem size. This is partially because with a large problem size, the GPU is fully used and more computations are required. Thus, the overhead in kernel launching is much smaller comparing with the time spend inside the kernel.
2.6.3 Multi-GPU Implementation and Speedup

One major contribution of this research is that it explores the new multi-GPU computing platform for CFD. This research extended CUFLow’s capability by letting it use multiple GPUs simultaneously. This implementation solved the issue of insufficient memory when the problem size grows larger. It also further reduces the computation time by a factor of 3.3 when using 4 GPUs.

To take advantage of multiple GPUs, the entire computational domain is first decomposed into several small sub-domains and each GPU holds one sub domain as shown in Figure 2.8. The surrounding red cells are ghost cells that are used for implementing boundary conditions. After domain decomposition, additional layers are required to hold data coming from the neighbor sub-domains as shown in Figure 2.9. LO/RO is the left/right outer halo layer, which is used to store the data coming from neighbor sub-domain. LI/RI is the left/right inner halo layer whose data will be transferred to its left neighbor. IN is the interior data which

Figure 2.8: Domain decomposition illustrated in 2D with 4 GPUs

...
The data transfer process between the GPUs are illustrated in Fig. 2.10 which shows the three steps required for transferring data from node 0 to node 1. In the first step, a CUDA memory copy command is issued to copy data from node 0 GPU memory to node 0 CPU memory. Here, assuming a distributed memory system, each node contains one CPU and one GPU, and GPU cannot directly talk with his neighboring GPU. Note, that in some platforms GPU can directly communicate with its neighboring GPU through the Remote Direct Memory Access (RDMA) [41], but this is only limited to some supercomputers with infiniband architecture [42]. Therefore, in the next step the Message Passing Interface (MPI) [43, 44] is used to transfer data cross different nodes. MPI send and receive commands move the data from node 0 CPU to node 1 CPU. After node 1 receives the data from node 0, another CUDA memory copy command moves the data from CPU 1 memory to its GPU memory to finalize the data transfer.

The data structures are also shown in Figure 2.10. In current version, the computational domain is decomposed only in \( z \)-direction. On each GPU, there are one extra layers of halo cells on each side of the \( z \)
boundary which are used to store the data from the neighbor GPU. When data transfer is required, the data in the inner halo layer is copied and transferred to its neighbor CPU’s outer halo layer as the arrows show in Figure 2.10. During the initial development of the multi-GPU solver, a four GPU work station was built using 4 C2075 GPUs. To use one CPU to control the four GPUs, a virtual machine was setup as shown in Figure 2.11. In this case, each GPU is assigned to one core of the Intel Xeon E5-2650v2 CPU. The host (CPU) memory is also assumed to be distributed to each core. Then during data communication, required data is first moved from the GPU memory to the assigned host memory. Then, each core plays the role as one MPI rank and uses the MPI send and receive calls to move the data. This enables the use of the multi-GPU code on both shared and distributed memory system.

The speed up of the code has been tested on an in-house four GPU workstation as well as on Blue Waters (BW) super computer. A 3D lid-driven cavity problem was first selected to test the performance of the multi-GPU code. The Navier-Stokes equations were solved on a grid of $128 \times 128 \times 512 \,(\sim 8\,\text{million})$ cells using the CPU version (paralleled with MPI) and multi-GPU versions. The times taken for advancing one flow timestep are compared in Figure 2.12(a). This comparison shows the GPU can significantly speed up the calculation and multiple GPUs can further speed up the solution procedure by factors between 3 and 4. In Figure 2.12(b), the stream traces of the flow in the center plane shows a big vortex with two small vortices at the bottom corners. Figure 2.12(c) shows the $v$ velocity in the center plane of the cavity and the boundary (vertical black lines) between the decomposed computational domain. This code was validated and used to study the three-dimensional flow in a driven cavity subjected to an external magnetic field, more details is available in Chapter 4 and Jin et al. [45].
2.7 References


[41] C. NVidia, C programming guide version 4.0, NVIDIA Corporation, Santa Clara, CA.


Chapter 3

Modeling and Measurements of Multiphase Flow and Bubble Entrapment

As introduced in Chapter 1., in steel continuous casting, argon gas is usually injected to prevent clogging, but the bubbles also affect the flow pattern, and may become entrapped to form defects in the final product. To investigate this behavior, plant measurements were conducted and a computational model was applied to simulate turbulent flow of the molten steel and the transport and capture of argon gas bubbles into the solidifying shell in a continuous slab caster. First, the flow field was solved with an Eulerian $k-\epsilon$ model of the steel, which was two-way coupled with a Lagrangian model of the large bubbles using a Discrete Random Walk method to simulate their turbulent dispersion. The flow predicted on the top surface agreed well with nailboard measurements, and indicated strong cross flow caused by biased flow of Ar gas due to the slide-gate orientation. Then, the trajectories and capture of over two million bubbles (25 $\mu$m to 5mm diameter range) were simulated using two different capture criteria (simple and advanced). Results with the advanced capture criterion agreed well with measurements of the number, locations, and sizes of captured bubbles, especially for larger bubbles. The relative capture fraction of 0.3% was close to the measured 0.4% for 1mm bubbles, and occurred mainly near the top surface. About 85% of smaller bubbles were captured, mostly deeper down in the caster. Due to the biased flow, more bubbles were captured on the inner radius, especially near the nozzle. On the outer radius, more bubbles were captured near to narrow face. The model presented here is an efficient tool to study the capture of bubbles and inclusion particles in solidification processes. All the figures and results presented in this chapter have been published[1].

3.1 Introduction

Argon bubbles captured during the continuous casting of steel are a major cause of defects, such as blisters and slivers, in rolled steel products. Ar gas is usually injected at the slide gate or stopper rod to prevent nozzle clogging. [2–4] The jets of molten steel then carry those bubbles through the Submerged Entry Nozzle (SEN) and into the mold cavity region, where they greatly affect the flow pattern, surface level fluctuations, and slag entrainment. Large bubbles captured near the surface can lead to blister defects, such as pencil
pipe, after rolling and annealing. [5, 6] Furthermore, the moving Ar bubbles collect nonwetting inclusion particles, such as alumina. If such a bubble is captured by the solidifying steel shell, the layer of inclusions covering its surface will lead to large oxide clusters, which cause severe sliver defects in the final product. [7, 8] Ar bubbles entering the mold region end up at three locations: (1) some reach the top surface, pass through the slag layer and escape harmlessly to the atmosphere; (2) some are captured near the meniscus and lead to surface defects; (3) some are captured deep in the caster and cause internal defects.

Many previous works have studied two-phase flow of argon and molten steel in the SEN and mold region of the continuous caster using water models and computational models. [3, 7–22] Increasing Ar gas causes increased upward flow near the SEN and tends to reverse the classic double-roll flow pattern to single-roll with surface flows away from the SEN towards the narrow face. [9, 17, 18] Asymmetric, oscillating flow is observed if gas fractions are excessive. [3, 23] Computational models of this multiphase flow should be three-dimensional and two-way coupled, as the Ar gas affects the steel flow and vice versa, especially with large gas fractions. [9, 10, 19]

Many researchers have used Eulerian-Eulerian flow models to investigate this two-phase flow problem. [9, 16–18, 24] Liu et al. [19] recently used inhomogeneous Multiple Size Group (MUSIG[25]) Eulerian models of the gas phase to describe the polydisperse bubbly flow in this process. Flow is affected by the input bubble size distribution. Liu et al. measured Sauter mean diameters (1-3 mm) in their physical water model, which are typical of previous measurements. [26]. Increasing gas flow generates more and larger bubbles. [2, 23, 26] Argon bubbles in steel are reported to be larger than air bubbles in water. [19, 26] Other models have used Lagrangian descriptions for the argon gas bubbles [9–13, 21, 27] such as the Discrete Particle Method (DPM) to study flow and bubble transport in the SEN and mold region. [10, 12, 14, 15, 27] Liu et al. [27] performed such Eulerian-Lagrangian two-way coupled $k\varepsilon$ simulations of Ar-steel flow with a coarse 0.15-million cell mesh. In the mold region, large bubbles (0.25-2.5 mm diameter) float directly upwards to the top surface upon exiting the SEN, while bubbles smaller than 0.25 mm travel with the jet across the mold cavity. A simple Eulerian-Eulerian model with a single equation set for the bubble phase misses this effect.

Relatively few previous flow studies have been extended to the important topic of motion and capture of particles, such as bubbles and inclusions.[10–15, 20–22, 27–29] The chaotic motion of individual particles in turbulent flow is automatically simulated by integrating the trajectories of large numbers of individual particles, as a post-processing step after Large Eddy Simulations of the transient flow field on a refined mesh. [12, 15, 30] With steady Reynold-Averaged Navier Stokes (RANS) models, this particle dispersion behavior can be mimicked using the random walk method. [10, 11, 13–15, 20–22, 27–29]
To predict the capture of Ar bubble and/or inclusion particles, a suitable capture criterion is needed. Many previous works have investigated the behavior of a particle at a solidification front [31–38] including fundamental numerical studies with simplified conditions such as planar-front solidification [31–33, 35–38] and initially stationary flow. Particles are pushed by the solidification front, unless a critical front speed for particle capture is exceeded. [31–38] However, this criterion is not suitable for direct application in continuous casting because the solidification front (1) is dendritic; and (2) experiences strong cross flow. The simple capture criterion that particles are entrapped if they touch the solidification front is often used in continuous casting. [12–14, 29, 30] This is a reasonable first approximation when the particle diameter is smaller than the PDAS. [12] When solidification heat transfer and the solid shell are included in the model, simulated particles can be automatically captured when their velocity reaches that of the solid shell. [14, 22] However, this method requires a very fine mesh of the mushy region, is very computationally intensive, and needed empirical fitting parameter(s) to match measurements. [14] Yuan, Mahmood and Thomas [13, 21, 30] developed a more advanced capture criterion, based on a local force balance on the bubble/particle at the solidification front, which includes the effect of particle size, Primary Dendrite Spacing (PDAS), interfacial tension and other effects. This criterion was applied here to study the capture of Ar bubbles in continuous casting process.

Several studies [10, 12–15, 20, 22, 28–30] have investigated the capture fraction and distribution of inclusion particles in continuous casting. Early studies of inclusion transport found that large inclusions (> 0.64mm) were likely to be entrapped just below mold [20, 28] while most small particles (≤0.04 mm) were trapped deep in the caster [12, 20, 28]. LES of flow in a thin-slab caster with no argon gas, with the “simple” capture criterion, only 8% of all inclusions (< 40 µm) were predicted to be removed. [12] The removal fraction was independent of both particle size and density, indicating that the particles were too small to deviate significantly from the flow trajectories of the fluid transporting them. [12] Using a $k – \epsilon$ flow model, including solidification heat transfer and the solid shell, with the simple capture criterion, Wang and Zhang [11] predicted the transport and entrapment of small inclusions in the full length of a billet caster, most 5 µm inclusions were captured deep in the strand. Using flow fields obtained from both RANS and LES simulations, with the advanced capture criterion, Thomas et al. [15] investigated the entrapment of larger particles (> 40 µm) including slag inclusions, alumina cluster, and bubbles during slab casting, and predicted overall particle removal fraction less than 20 pct. Gas injection was found to increase the removal rate (average number of inclusions removed per unit time) of small particles. [15] Also, to achieve reproducible results, (removal rate variations within ±3%), > 2500 particles of each size should be tracked. [15] None of these studies of particle transport included the coupled effects of Ar gas on the flow
This chapter investigates the transport and capture of argon bubbles in a real commercial continuous caster using both the simple and advanced (force balance criterion [13, 15, 30]) capture criteria. The results are compared with plant measurements for verification.

### 3.2 Plant Measurements

Plant measurements of fluid flow and bubble entrapment were conducted at Baosteel Shanghai in 2012 on the No. 4 continuous slab caster, which casts conventional (230mm thick) slabs. The flow rate of the molten steel through the SEN into the mold is controlled by a slide-gate that moves between the geometric center and the Inside Radius (IR) side of the caster. For the experiments casting 1300mm wide slabs at 1.5m/min, the slide gate was 70% open, as shown in Figure 3.1. Argon gas was injected through porous refractory in the upper tundish nozzle and SEN to prevent clogging, not including Ar gas provided to the slide gate and upper plate seal regions to avoid air aspiration. Top surface velocities were measured with two sets of nailboard dipping tests, which are widely used to measure mold surface flow. [12, 39] Specifically, velocities were extracted from the height differences of solidified lumps according to the equation in figure 4 of Ref. [39] Casting conditions and process parameters are given in Table 3.1. Previous work showed that the gas heats up to the steel temperature before it enters the nozzle, thus the “hot” argon flow rate is used in the simulation. The equation to convert cold (room temperature and pressure of 1 atm) to hot argon volume fractions are available elsewhere. [3, 9]

To investigate the capture of Ar bubbles during the process for the same conditions in Table 3.1, samples were cut from the center and quarter of the Wide Face (WF), and Narrow Face (NF) of the steel slab. The

![Figure 3.1: Slide gate configuration schematic](image)

![Figure 3.2: Locations of samples and six layers examined on each](image)
Table 3.1: Process parameters

<table>
<thead>
<tr>
<th>Process parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mold thickness ( (L_t) )</td>
<td>230 mm</td>
</tr>
<tr>
<td>Mold width ( (L_w) )</td>
<td>1300 mm</td>
</tr>
<tr>
<td>SEN Submergence depth</td>
<td>160 mm</td>
</tr>
<tr>
<td>Nozzle port downward angle</td>
<td>15°</td>
</tr>
<tr>
<td>Nozzle port area (width ( \times ) height)</td>
<td>65×83 mm(^2)</td>
</tr>
<tr>
<td>Casting speed ( (V_c) )</td>
<td>1.5 m/min</td>
</tr>
<tr>
<td>Argon volume fraction ( (\alpha) )</td>
<td>8.2% (hot)</td>
</tr>
<tr>
<td>Steel density ( (\rho) )</td>
<td>7000 kg/m(^3)</td>
</tr>
<tr>
<td>Argon density ( (\rho_p) )</td>
<td>0.5 kg/m(^3)</td>
</tr>
<tr>
<td>Steel viscosity ( (\mu) )</td>
<td>0.0063 kg/(m·s)</td>
</tr>
<tr>
<td>Ar viscosity ( (\mu_p) )</td>
<td>0.0000212 kg/(m·s)</td>
</tr>
<tr>
<td>Sample length in casting direction ( (\Delta z) )</td>
<td>150 mm</td>
</tr>
</tbody>
</table>

To quantify the number and location of the bubbles captured in the samples, the outer 3 mm of the surface of each sample was milled away, and then a 35× optical microscope was used to record the diameter of each bubble observed on the exposed surface, and the number of bubbles in each size range was counted. After recording the results, another 3mm steel was milled away and the bubbles on the new exposed surface were measured. This procedure was repeated to examine six layers total, denoted \( s_j \), located at 3, 6, 9, 12, 17 and 22 mm beneath the outer surface of the slab NF or WF. Figure 3.3 shows a SEM micrograph of a large bubble captured in one of the samples. Note the many small inclusion particles that cover the surface.

Figure 3.3: Example of large bubble \( (d_p = 1.45 \, \text{mm}) \) found in plant measurements

of the hollowed-out roughly-hemispherical depression that comprises most of the bottom half of the original bubble.
3.3 Computational Models and Solution Procedure

A three-dimensional finite-volume computational model together with Lagrangian tracking method was applied to study the fully-coupled turbulent flow behavior and the transport phenomena of Ar bubbles in a commercial continuous steel caster. The computational domain includes half of the slide gate, SEN and mold region (from meniscus surface to 2.5 m below meniscus), as shown in Figure 3.4. First, steady-state solution of single-phase flow of molten steel was obtained by using standard $k-\epsilon$ model. A realistic bubble-size distribution was selected and discretized. Then, starting with the single-phase flow field, RANS and Lagrangian Discrete Phase Model (DPM) coupled simulations were conducted to predict a pseudo steady flow field solution including the effects of Ar gas injection. The Discrete Random Walk (DRW) model was applied to include the effect of turbulence on bubble dispersion. After that, the flow field was fixed, and about 2.5 million bubbles were injected at the SEN inlet to study the transport phenomena and the capture of the Ar bubbles. Two different capture models were implemented and compared. The entire solution procedure is shown in Figure 3.5.

![Figure 3.4: Mesh of the computational domain of the slide gate, SEN, and mold region](image)

3.3.1 Initial Flow Model

A single-phase steel flow simulation was first carried out to provide an initial guess for the later two-way coupled steel-Ar flow simulation. Solid shell is included in the domain and the shell thickness can be determined by $s = 3\sqrt{t}$ mm. The solid shell (necessary for later study of magnetic field effects) is modeled as a solid zone in the solver [40] and therefore the fluid flow equations were not solved in the shell region.
The continuity and momentum equations for this RANS model are:

\[
\frac{\partial}{\partial x_i} (\rho U_i) = S_{\text{mass-sink}} \tag{3.1}
\]

\[
\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + S_i_{\text{momentum-sink}} \tag{3.2}
\]

Mass and momentum sink (source) terms [30, 41, 42] were added at cells adjacent to the liquid-solid interface to include the effect of fluid flow across the domain boundary due to solidifying of the shell. The mass sink \(S_{\text{mass-sink}}\) is calculated as:

\[
S_{\text{mass-sink}} = \rho v_c A_z V_{\text{cell}}^{-1} \tag{3.3}
\]

where \(A_z\) is the projected shell area of the cell in the casting direction and \(V_{\text{cell}}\) is the volume of each (red) cell, as illustrated in Figure 3.6. The momentum sink is the loss of the momentum associated with the mass loss, and is evaluated as follows:

\[
S_i_{\text{momentum-sink}} = \rho v_c A_z V_{\text{cell}}^{-1} U_i \tag{3.4}
\]

Figure 3.6: Computation of mass sink term for shell solidification

The \(k - \epsilon\) model was used to model turbulence. [43, 44] The turbulent viscosity \(\mu_t\) needed is defined as \(\mu_t = \rho C_{\mu} k^2 \epsilon^{-1}\). It can be found by solving the following transport equations for \(k\) and \(\epsilon\):

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k U_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_j} \right] - \mu_t S^2 - \rho \epsilon \tag{3.5}
\]

\[
\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon U_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (\mu_t S^2) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} \tag{3.6}
\]
where $S = \sqrt{2S_{ij}S_{ij}}$ is the modulus of the mean rate-of-strain tensor. The constants in these equations are $C_{1e} = 1.44$, $C_{2e} = 1.92$, $C_{\mu} = 0.09$, $C_k = 1.0$ and $C_\epsilon = 1.3$.

At the slide gate inlet, fixed velocity $V_{inlet} = 1.69$ m/s was applied, based on steel volume flow rate $(0.007475 \text{ m}^3/\text{s})$ divided by inlet area $(0.0044 \text{ m}^2)$. The turbulent kinetic energy and its dissipation rate were assumed to be small as $10^{-4} \text{ m}^2/\text{s}^2$ and $10^{-5} \text{ m}^2/\text{s}^3$, respectively. A pressure outlet boundary condition was applied at the domain bottom based on the ferrostatic pressure of steel (171.5 kPa) at that distance (2.5 m below the top surface) multiplied by the steel density and gravity constant. The turbulent kinetic energy and its dissipation rate were specified as $10^{-5} \text{ m}^2/\text{s}^2$ and $10^{-5} \text{ m}^2/\text{s}^3$ for reverse flow from the bottom boundary, respectively. A free slip boundary condition was applied at the top surface of the mold. The WF and NF solidification fronts and SEN walls had standard wall laws. [44]

### 3.3.2 Bubble Size Distribution Model

In this study, the Ar gas volume was distributed into bubbles according to a Rosin-Rammler [45] size distribution, which was originally used to describe solid particle distributions. The ideal Rosin-Rammler cumulative distribution $F(d)$ is defined by the mean diameter $d_{mean}$ and spread parameter $\eta$:

$$F(d_i) = \frac{V_g(d<d_i)}{V_g} = 1 - \exp \left(-\frac{d_i}{d_{mean}}\right)^\eta$$

(3.7)

In this work, a discrete form of this distribution function was based on 11 different bubble sizes, $d_i$, where $i$ is the bubble size group, and $F(d_i)$ is the volume fraction of Ar contained in those bubbles with diameter less than $d_i$.

The average bubble diameter $d_{mean}$ was taken as 3 mm, based on calculations with a previously-validated two-stage model of Ar injection into downward flowing steel [3, 26] for the flow rates used in this work. This value also matches recent work [19] based on water model measurements using a MUSIG model. The spread parameter $\eta$ was taken as 4, based on a measurement in a water model [46] and adjusted to account for increased surface tension in steel/argon using a relation from previous measurements [3, 26] and recent measurements [47] of bubble distributions. As the size distribution used in the model agrees with measured bubble size in the mold region, the effects of pressure on bubble size is not included in the model.

The volume fractions of bubbles with different diameters are given in Figure 3.7, where different definitions of the mean bubble diameters are noted. The blue squares represent the diameter and volume fraction of each group of bubbles of the total Ar volume used in the model. The red stair-case-like line is the summation over the volume fraction of bubbles with diameter less than the specified diameter. This staircase Rosin-
Rammler cumulative distribution is compared with the ideal distribution, plotted as a dashed line, which overlaps at discrete points. Note also that bubbles smaller than 1 mm in diameter comprise less than 1% of the total bubble volume.

![Figure 3.7: Ar bubble volume fraction distribution](image)

3.3.3 Two-phase Flow Model

After the single-phase fluid flow solution was obtained, the effect of the Ar gas bubbles on fluid flow was added. The Eulerian \( k - \epsilon \) flow model with Lagrangian DPM tracking and DRW model was used to include this phenomenon. In this two-way coupled Eulerian-DPM model, an additional source term \( S_{DPM} \) is added to equation (3.2), to include the net force from the Ar bubbles to the local fluid. The following force balance equation was solved for each individual bubble with mass \( m_p \) and velocity \( u_p \):

\[
m_p \frac{du_p}{dt} = \frac{m_p 18 \mu C_D}{\rho_p d_p^2} \left( u - u_p \right) \rho d_p \frac{u_d - u}{\mu} + 0.5m_p \frac{\rho}{\rho_p} \left( \frac{Du}{Dt} - \frac{du_p}{dt} \right) + m_p \frac{\rho}{\rho_p} \frac{Du_p}{Dt} + m_p \frac{g (\rho_p - \rho)}{\rho_p} \tag{3.8}
\]

where the four forces are: drag \( F_D \), virtual mass \( F_V \), pressure gradient effect \( F_p \) and buoyancy/gravity \( F_b \). The drag force depends on particle Reynolds number \( Re_p \) and the drag coefficient \( C_D \) was computed based
on the drag law proposed by Morsi and Alexander [48]:

\[ C_D = a_1 + \frac{a_2}{\text{Re}_p} + \frac{a_3}{\text{Re}_p^2} \] \hspace{1cm} (3.9)

where \( a_1, a_2, a_3 \) are given by:

\[
a_1, a_2, a_3 = \begin{cases} 
0, 24, 0 & 0 < \text{Re}_p < 0.1 \\
3.690, 22.73, 0.0903 & 0.1 < \text{Re}_p < 1 \\
1.222, 29.1667, -3.8889 & 1 < \text{Re}_p < 10 \\
0.6167, 46.50, -116.67 & 10 < \text{Re}_p < 100 \\
0.3644, 98.33, -2778 & 100 < \text{Re}_p < 1000 \\
0.357, 148.62, -47500 & 1000 < \text{Re}_p < 5000 \\
0.46, -490.546, 578700 & 5000 < \text{Re}_p < 10000 \\
0.5191, -1662.5, 5416700 & \text{Re}_p > 10000 \\
\end{cases} \hspace{1cm} (3.10)
\]

The first 3 of these forces comprise \( S_{DPM} \). To save computation time and considering that the volume fraction \( d_p < 1 \text{ mm} \) bubbles is so small, the effect of these small bubbles on the mold flow pattern in this two-way coupled simulation was negligible. Therefore, only the large bubbles (1-5 mm) were injected and tracked in this step.

To include the effect of turbulence on particle dispersion, a DRW model was used,

\[ u_i = U_i + u'_i \approx U_i + \xi \sqrt{2k/3} \] \hspace{1cm} (3.11)

where the local fluid velocity \( u_i \) is a function of local time averaged velocity \( U_i \), local turbulent kinetic energy \( k \) and a normally-distributed random number \( \xi \) with mean 0 and standard deviation 1. During the calculation, \( \xi \) changes to produce a new instantaneous velocity fluctuation whenever the time reaches the smaller of: the eddy lifetime \( t_e \) and the time needed to cross the eddy \( t_{cross} \), which are defined as follows:

\[ t_e = -0.15 \frac{k}{\epsilon} \ln (\gamma) \] \hspace{1cm} (3.12)

\[ t_{cross} = -\tau_p \ln \left[ 1 - \frac{C \mu \frac{v}{k} \frac{\epsilon}{\mu} \frac{1}{\epsilon}}{\tau_p |u - u_p|} \right] \text{ where } \tau_p = \frac{\rho_p d_p^2}{18 \mu} \] \hspace{1cm} (3.13)

where \( \gamma \) is a random number uniformly distributed from 0 to 1, and \( \tau_p \) is the particle relaxation time which...
represents a time scale for a particle to respond to changes in the surrounding flow. The velocity and position of each particle is computed and updated until it escapes or is captured.

### 3.3.4 Bubble Tracking and Capture Model

After solving for the steady state flow fluid using the Eulerian-Lagrangian model, particles were injected into the domain and their trajectories were tracked. The number of bubbles injected with diameter $d_i$, denoted $N(d_i)$, is determined to match the plant experiment conditions as follows:

$$
N(d_i) = \frac{3\alpha}{4\pi(0.5d_i)^3},
$$

where

$$
\alpha = \begin{cases} 
F(d_i) & \text{if } i = 1 \\
F(d_i) - F(d_{i-1}) & \text{if } i > 1
\end{cases}
$$

(3.14)

where $V_g$ is the volume of total Ar gas injected into half of the caster during the chosen time $t_{total}$ of 60s; $L_t$ and $L_w$ are the strand thickness and width, respectively; and $\alpha$ is the total Ar volume fraction of 8.2%. The volume fractions for bubbles of different sizes are denoted as $\alpha_i$, and can be determined based on $F(d_i)$ from equation (3.7). Following these equations, ~2.5 million bubbles were injected over 60s and tracked for each capture criterion.

Two different capture criterion were implemented in this work: (1) a “simple” capture criterion which assumes immediate capture when a bubble/particle touches the solidification front; (2) an “advanced” capture criterion that based on the force balance proposed by Yuan and Thomas. [13, 21, 30] The flow chart in Figure 3.8 (reprint from Ref. [15]) shows the procedure of this advanced capture criterion. For small bubbles less than the PDAS, the particle can enter between the dendrite arms and be captured by entrapment. For bubbles/particles greater than the PDAS, the advanced criterion considers eight forces acting on a spherical bubble/particle touching three dendrite arms, shown in Figure 3.9 (reprint from Ref. [15]). In addition to the four forces shown in equation (3.8), lift force $F_L$, lubrication force $F_{lub}$, Van der Waals force $F_{IV}$, and surface tension gradient force $F_{Grad}$ were also considered [13, 15, 21, 30]:

$$
F_L = -\frac{9}{4\pi \mu d^2 U_s} \epsilon \sqrt{G} \left( \frac{G}{v} \right)^{1/2}, \quad G = \frac{du_1}{dy}, \quad \epsilon = \text{sgn}(G) \frac{\sqrt{v^G}}{U_s}, \quad U_s = u_1 - v_1
$$

(3.15)

$$
J(\epsilon) = 0.6765 \left[ 1 + \tanh \left( 2.5 \log_{10} \epsilon + 0.191 \right) \right] \left[ 0.667 + \tanh \left( 6\epsilon - 1.92 \right) \right]
$$

$$
F_{lub} = 6\pi \mu V_{sol} \frac{R_p^2}{h_o} \left( \frac{r_d}{r_d + R_p} \right)^2
$$

(3.16)
Particle contacts a boundary representing mushy-zone front? (yes/no)

Particle size larger than PDAS ($d_p > \text{PDAS}$)? (no)

In solidification direction, repulsive force smaller than attractive force? $F_t - F_{\text{attractive}} > 0$ (yes/no)

Can cross-flow and buoyancy drive particle into motion through rotation? (yes/no)

$F_{\text{cross}} + F_{\text{attractive}} \cos \theta + (F_t - F_{\text{attractive}}) \sin \theta > (F_{\text{attractive}} - F_t) \sin 2\theta$, if $F_{\text{cross}}$ and $F_{\text{attractive}}$ in the same direction

or

$F_{\text{cross}} - F_{\text{attractive}} \cos \theta + (F_t - F_{\text{attractive}}) \sin \theta > (F_{\text{attractive}} - F_t) \sin 2\theta$, if $F_{\text{cross}}$ and $F_{\text{attractive}}$ in opposite direction

and $F_{\text{cross}} \geq F_{\text{attractive}}$

or

$F_{\text{cross}} - F_{\text{attractive}} \cos \theta + (F_t - F_{\text{attractive}}) \sin \theta > (F_{\text{attractive}} - F_t) \sin 2\theta$, if $F_{\text{cross}}$ and $F_{\text{attractive}}$ in opposite direction

and $F_{\text{cross}} < F_{\text{attractive}}$

Figure 3.8: Flow chart for Ar bubble/particle capture criterion

Figure 3.9: A Bubble/Particle Touching 3 Dendrite Tips
\[ F_{IV} = 2\pi \left( E_{sp} - E_{sl} - E_{pl} \right) \frac{r_d R_p}{r_d + R_p} \frac{a_0^2}{h_0^2} \]  

(3.17)

\[ F_{Grad} = -\frac{m \beta \pi R_p}{\xi^2} \left\{ \frac{\xi^2 - R_p^2}{\beta} \ln \left( \frac{\xi + R_p}{\alpha (\xi - R_p) + \beta} \right) + \frac{2R_p}{\alpha} \frac{\beta}{\alpha^2} \ln \left( \frac{\alpha (\xi - R_p) + \beta}{\alpha (\xi + R_p) + \beta} \right) \right\} + \frac{2R_p}{\alpha} - \frac{\beta}{\alpha^2} \ln \left( \frac{\alpha (\xi - R_p) + \beta}{\alpha (\xi + R_p) + \beta} \right) \]  

(3.18)

where \( u_1 \) and \( v_1 \) are instantaneous streamwise velocities for the liquid and particle; \( R_p \) is the radius of the particle; \( r_d \) is the radius of the dendrite tip; \( h_0 \) is the distance between the particle and the dendrite tip; \( a_0 \) is the atomic diameter of liquid steel; \( E \) is the surface energy and the subscripts \( s, p \) and \( l \) denote the solid steel, particle/bubble and liquid steel respectively; \( \xi = R_p + r_d + h_0 \), \( \alpha = 1 + n C_0 \) and \( \beta = (C^* - C_0) n r_d \) where \( m, n, C^* \) and \( C_0 \) are model parameters. More details about the formulation of these forces can be found in previous work. [13, 15, 30] Note that bubbles reaching the bottom of the domain were also considered as captured.

The captured bubbles were then analyzed to compute the capture rate and capture fraction for different size of bubbles for both capture criteria. The captured numbers of bubbles and their diameters were recorded, and the capture fractions for bubbles of different diameters were denoted as calculated using:

\[ (d_i) = \frac{n(d_i)}{N}\]  

(3.19)

where, \( n(d_i) \) denotes the capture rate, or number of captured bubbles of diameter \( d_i \) in the entire caster during a chosen time interval.

### 3.3.5 Model to Predict Sample Observations

To compare with plant measurements, the first step is to determine the number \( n_{sample}(d_i) \) of each bubble size captured during the time the samples were cast, which is determined as follows:

\[ n_{sample}(d_i) = n(d_i) \frac{t_{total}}{t_{sample}} \]  

(3.20)

where \( t_{total} \) is 60s in this study and \( t_{sample} = V_c/\Delta z \), is 6s in this study.

Next, the bubbles that can be observed on each sample surface must be extracted from these results. As illustrated in Figure 3.10, a bubble \( k \) with radius \( r_k \) captured at a distance \( h_k \) beneath the strand surface, can be observed on surface \( j \) located at distance \( s_j \) beneath the outer slab surface (WF/NF) only if it satisfies
one of the following conditions:

\[ \{ h_k < s_j \text{ and } h_k + r_k > s_j \} \text{ or } \{ h_k > s_j \text{ and } h_k - r_k < s_j \} \]

\[ (3.21) \]

\[ d_{\text{true}} = d_{\text{visible}}/0.785 \]

(3.22)

These conditions were checked for each bubble, to extract the number and average diameter of bubbles predicted to be observed on each examined sample layer. The polished sample surface is unlikely to cut through captured bubbles at their largest center diameter. Thus, the observed circle diameters are usually smaller than the real sphere diameters, as illustrated in Figure 3.11. A simple method proposed by Lekakh et al. [49] was used here to convert the observed diameters into realistic diameters of the spherical bubbles, which are presented in the results.

\[ \text{Figure 3.10: Determine the number of bubbles that can be observed on surface} \]

\[ \text{Figure 3.11: Convert hole diameter observed on sample surface to true bubble diameter} \]

### 3.3.6 Computational Details

The above models were implemented into the commercial finite-volume package ANSYS FLUENT [40] and solved on the structured mesh of \( \sim 1 \) million hexahedral cells shown in Figure 3.4, according to the flow chart in Figure 3.5. The mass and momentum sinks and the advanced capture criterion were added through a User Defined Function (UDF). The simulations were run in parallel on a six-processor Intel Xeon X5650 CPU (2.66 GHz). First, the single phase model was run for the initial condition, which took about one day of CPU time. Then, about three days were needed for the two-phase fluid flow solution. Then, about one day CPU time was needed to track the \( \sim 2.5 \) million bubbles for each capture criterion. Finally, a MATLAB
code was used to process the output data to extract the predicted distributions in the sample, which took \( \sim 3 \) hours to run.

3.4 Model Results and Discussion

3.4.1 Fluid Flow in SEN and Mold

Contours of steel flow speed and velocity vectors on the symmetry plane of the SEN and slide gate are shown in Figure 3.12. Because the slide gate moves towards the IR, the steel flow down that side of the SEN is stronger. This leads to a counter-clockwise eddy or “swirl” at the SEN bottom region. This swirl extends completely through the port as shown in Figure 3.13. Consequently, a higher Ar volume fraction region (gas pocket) is seen at the top and IR side of the port, where more Ar bubbles escape. This gas pocket matches observations in water models, although the calculation is not exact, owing to local mass imbalance inherent to the computational method in regions of high gas fraction.

Contours of steel velocity magnitude and streamlines in the middle plane of the caster mold are shown in Figure 3.14. The standard double-roll flow pattern is modified by the argon injection and three eddies (recirculation regions) were observed (labeled 1,2,3). The nozzle jets impinge on the narrow face, and split, sending some recirculating flow upwards and across the top surface towards the SEN. This is met by flow rising up beside the SEN driven by the buoyancy of the argon gas. The asymmetric swirl caused by the slide gate sends more gas up the inner radius (IR) of the WF. Contours of Ar volume fraction and steel velocity

![Figure 3.12: Ar volume fraction and velocity field in symmetry plane of SEN](image-url)
Figure 3.13: Ar volume fraction and steel vector field at port outlet

Figure 3.14: Velocity magnitude and streamlines in middle plane
vectors at 1 cm below the top surface are shown in Figure 3.15. The surface flow pattern is separated into two zones by the purple dashed line. Surface flow near the SEN is driven by eddy 2 in Figure 3.14 and is caused by the high buoyancy of the flow containing high Ar gas fraction. The opposite flow from the NF is driven by eddy 3 in Figure 3.14 from flow up the NF wall. The upward flow near the SEN is biased towards the IR side of the mold wall, due to the port swirl caused by the slide-gate movement towards the IR. This Ar-rich flow upwells on the top surface from the IR, causing strong cross flow across most of the top surface towards the OR. The maximum cross flow velocity reaches \( \sim 0.3 \text{ m/s} \).

![Figure 3.15: Velocity and Ar volume fraction near top surface](image)

It also important to mention that this biased flow is not seen in the single phase simulation step. Therefore, this biased flow pattern is induced by the combined effect of asymmetric slide-gate movement and the Ar gas injection. Without Ar gas, the asymmetric effect caused by the slide gate is only important in the SEN and the swirl exiting the ports dissipates in the mold. With enough Ar injection, the slide gate movement causes the important effect of detrimental asymmetric cross flow on the top surface.

### 3.4.2 Bubble Capture Locations

The trajectories and capture locations of 2.5 million Ar bubbles of different sizes were calculated based on the steady state flow solution, for both the simple and advanced capture criteria. The captured locations of the small \( (d_p < 0.1mm) \) and medium-sized \( (0.1 \leq d_p \leq 0.3mm) \) bubbles are shown in Figure 16 and Figure 17, for the simple and advance capture criterion respectively. The horizontal black lines show the location of the sample surfaces examined in the measurements.

All figures show that the number of bubbles captured by the solidifying shell decreases with distance down the caster. However, many small bubbles are captured deep in the caster, which agrees with previous studies. \([14, 20, 27, 28]\) More bubbles are captured near the SEN on the IR WF, but on the OR WF more bubbles are captured near the narrow face, especially where the jet spreads to impinge on the solidification front. Many bubbles are captured on the narrow face in the mold near and above the point of jet impingement.
Comparing Figure 16 and 17 shows that 1.4X fewer 0.3 mm bubbles (red dots) were captured when using the advanced capture criterion.

Large captured bubbles ($d_p \geq 1mm$) are most detrimental to the final product. With the simple criterion, Figure 3.18 shows many large bubbles captured on the solidifying walls, especially on the IR, where many captured bubbles exceeded 3mm. Elsewhere, on OR and NF, the captured large bubbles were mainly $\leq 1$ mm. With the advanced capture criterion, Figure 3.19 shows that relatively few large bubbles ($\leq 2$ mm) were captured and mostly very close to the meniscus, which agrees with previous results.[20, 27, 28]

The locations where Argon bubbles escaped from the top surface are plotted in Figure 3.20. As expected, most large bubbles escape near the SEN, biased towards the IR side of the caster, where the gas fraction...
Figure 3.20: Bubbles escaped from top surface using simple (top) and advanced (bottom) capture criterion was highest in Figure 3.15. Figure 3.20 also shows that smaller bubbles are more evenly distributed over the surface. Relatively few small bubbles \((d_p < 0.1 \text{ mm})\) are seen escaping the top surface, which indicates that most small bubbles are captured in the cast product.

### 3.4.3 Bubble Capture Rate and Fraction

Figure 3.21 shows the capture rates of different Ar bubble sizes, comparing the simple and advanced capture criteria. The total injection rate of each bubble size is plotted with an open circle. To compare with the measurements, the capture rates were evaluated for 6 seconds (the time to cast the sample length \(\Delta z = 150 \text{ mm}\)), and summed over the entire caster. The predicted capture rates for bubbles less than 0.2 mm by the simple and advanced capture criteria are very close and were only slightly lower than the injection rate. In contrast, huge differences in capture rates were observed for larger bubbles. The simple criterion predicted round three thousands 1 mm bubbles captured in the entire caster during 6 seconds of casting process, while the advanced capture criterion predicted around 50 bubbles per 6 sec, which is \(60 \times\) fewer. With larger bubbles, the advanced criterion predicted even fewer (up to \(500 \times\) fewer). The capture rate dropped rapidly with increasing bubble diameter for both capture criteria.

It important to note that no bubbles greater than 3 mm diameter were captured when using advanced capture criterion or in the measurements. In the plant measurements, only one 1.4 mm bubble was observed in one of the examined layers. To compare with the model predictions, the measurements were scaled based on the peripheral of the caster. Specifically, only one 1.4mm bubble was observed in one of the examined layers of all of the sample layers, which extended 530 (150+150+230) mm of the total 3060 (1300+1300+230+230)
mm perimeter. This means that around 6 captured bubbles would be expected if the entire perimeter were examined, which is the number plotted as the scaled experiment. Note that even more than 6 bubbles should be found if the experiments examined the entire samples instead of only 6 slices. Considering this would increase the measurements closer to the 40 ∼1 mm bubbles predicted by the advanced criterion.

The fraction of bubbles that are captured is plotted in Figure 3.22 for different sizes. The capture fractions of small bubbles (any $d_p < 0.1\ mm$) is ∼0.85 for both capture criteria. The uniformity of this value for any small size shows that the capture fraction of bubbles smaller the PDAS is governed only by the flow pattern. However, the capture fractions of large bubbles decreases dramatically with increasing bubble size. The capture fraction of 1 mm bubbles was 6% with the simple capture criteria but only 0.1% with the advanced capture criterion. The advanced criterion predicts a drop of about one order of magnitude in capture fraction from 0.01% of 1 mm bubbles to 0.01% of 2 mm bubbles, to 0.001% of 3 mm bubbles, and no capture at all of bubbles larger than 3 mm.

### 3.4.4 Bubble Removal Fraction and Capture Distribution with Advanced Criterion

The removal fraction for different bubbles and the relation between bubble capture percentages and distance below meniscus are illustrated in Figure 3.23. The removal fraction for large bubbles ($1 \leq d_p \leq 5\ mm$) was 99.98%, for medium bubbles ($0.1 \leq d_p \leq 0.3\ mm$) was ∼48.5% and for small bubbles ($d_p < 0.1\ mm$) was ∼16.1%. Previous RANS and LES studies [9] showed removal rates for 0.1 and 0.4 mm slag particles of ∼13% and ∼47%, which is close to the present results. These Sankey diagrams also reveal that large bubbles are very rarely captured in the casting process. Around 39% of small bubbles and 28% of medium
Bubble capture fraction are 0 for bubbles $d_p > 3$ mm, with advanced capture criterion. Fraction = 0.85

Figure 3.22: Bubble capture fraction for different bubble diameters

Figure 3.23: Sankey diagram of number of bubbles captured in different zones for (a) large bubbles (b) medium bubbles and (c) small bubbles
bubbles are captured within 0.5 m of the meniscus, which is confined to the first 13 mm of shell growth in the present study. The trend of less capture with downward distance in the casting direction is also illustrated. This means less bubbles would be captured in the interior region of the slab. Below 0.5 m, the capture fraction of medium bubbles decreases to about half that of small bubbles. The average capture percentage for large bubbles was around 0.02% which is very small, and they are only be captured at the top region of the meniscus, which means very close to the strand surface.

The capture rates of different size ranges of bubbles at different locations are illustrated in Figure 3.24. Capture rates were calculated from the number of bubbles captured within a region (labeled from I to IV in Figure 3.24) divided by the casting time (6 seconds in this case) and the surface area of the region. Capture rates decrease with distance below the meniscus. The capture rates of medium bubbles were larger than both small and large bubbles in each region. Capture rates on the IR side were larger than on the OR side. This is due to more gas flow leaving the port towards the IR side, increasing bubble volume fraction near the IR. The measured number of bubbles captured per cm\(^2\) at different depths beneath the WF/NF are compared in Figure 3.25 with predictions using the advanced criterion. In the center region, Figure 3.25(a) shows that the predicted 0.05 to 0.1 bubbles/cm\(^2\) captured is close to the measurements of 0.05 to 0.2 bubbles/cm\(^2\). The predictions match better deeper beneath the strand surface, but under-predict capture significantly near the meniscus. This may be due to the neglect of hook capture mechanism in the current model. Meniscus solidification creates hooks which extend out from the solidification front and entrap rising bubbles into the shell. This mechanism occurs only near the meniscus before hooks are overtaken by the growing shell and no
longer affect capture. At the quarter regions, Figure 3.25(b) shows that more bubbles were predicted to be captured per cm$^2$ on the OR than the IR. The OR predictions matched better with the measurements. On the NF, Figure 3.25(c) shows the predicted 0.05 to 0.13 bubbles/cm$^2$ captured is close to the measurements of 0 to 0.17 bubbles/cm$^2$. Fewer bubbles are captured deeper in the slab.

Figure 3.25: Predicted number of bubbles captured per cm$^2$ at (a) center region, (b) quarter region and (c) NF region

### 3.5 Model Verification

The results of this work show reasonable agreement between the computational model predictions and the measurements.

#### 3.5.1 Surface Velocities

The speed profile across and just beneath the top surface centerline from the multiphase flow model are compared with the plant nailboard measurements in Figure 3.26. The strong cross flow predicted from IR toward OR agrees with plant measurements. The cross flow is due to the biased gas flow caused by the argon gas slide-gate moving towards the IR. The plant measurements are even stronger than the predictions. However, only two nailboard measurements of this chaotic flow were available, so perfect quantitative matching is not expected. Further measurements are needed to better quantify the steel-Ar flow pattern.

#### 3.5.2 Bubbles Captured in Samples

The predicted number and average diameter of bubbles captured on each examined surface of the three samples are compared against plant measurements in Figure 3.27 and 28. Figures 3.27(a) and (b) show the predicted and measured number of bubbles captured per 150×150 mm$^2$ sample layer of the center sample and the quarter sample, respectively. In the center sample, both criteria under-predict the number of captured
bubbles close to the strand surface, which means captured close to the meniscus region. At the quarter region, the advanced capture criterion predicted 20 to 40 bubbles on the first two layers of the examined surfaces, which matches the measurement of 35 bubbles. However, the simple criterion over-predicted the number of captured bubbles by $4 \times$, especially in the second layer of the quarter sample. Both criteria predict asymmetric capture on IR and OR. Figure 3.27(c) and (d) show the predicted and measured average bubble diameter (true diameters) on each examined sample layers of the center and quarter region samples. In the center region sample, the measured average bubble diameter was $\sim 0.1$ to 0.2 mm while the simple capture
criterion predicted 0.3 to 0.5 mm. The advanced criterion predicted 0.1 to 0.3 mm which is closer to the measurements (agrees better than that from simple capture criterion). In the quarter region sample, the measured average bubble diameters were 0.1 to 0.2 mm, while the simple criterion over-predicted (0.3 to 1.6 mm). The advanced capture criterion predicted 0.2 to 0.4 mm, which although not exactly matching the measurements, is much better than the simple criterion. The predicted numbers and average diameters of captured bubbles on each layer of the NF samples are shown in Figure 3.28. Both criteria under-predicted the number of captured bubbles close to the strand surface, and over-predicted their average diameter. However, the advanced criterion exceeded the measurements by $\sim 2$ times, while the simple criterion was almost 4 times larger.

![Graph showing predicted number of bubbles captured per sample layer and corresponding average bubble diameter](image)

Figure 3.28: Predicted (a) number of bubbles captured in each layer of NF sample and (b) corresponding average bubble diameter

### 3.5.3 Relative Capture Rate of Large Bubbles

In all of the measured sample layers, $\sim$500 bubbles were observed and only one large bubble (1.4 mm diameter) was observed $> 0.5 \text{mm}$. So, the fraction of all observed captured bubbles $> 1 \text{mm}$ is $\psi(1 \text{mm}) = 0.2\% \ (1/500)$. To predict this capture fraction for $> 1 \text{mm}$ bubbles, table 3.2 shows: the total number of bubbles injected into the domain during the particle tracking step (column 2); the number of 1 mm bubbles injected (column 3); the total number of bubbles captured in the entire caster (column 4); the number of 1 mm bubbles captured in the entire caster (column 5); the fraction of 1 mm bubbles that were captured (column 6), and the fraction of captured bubbles that were larger than 1 mm (column 7). These results show that the advanced capture criterion prediction of 0.3% matches closely with the plant measurement (0.2%). The simple criterion greatly overpredicts the measurements (by $33\times$).
Table 3.2: Capture fractions of 1 mm bubbles

<table>
<thead>
<tr>
<th>Criteria</th>
<th>$\Sigma N(d_i)$</th>
<th>$\Sigma N(1 \text{ mm})$</th>
<th>$\Sigma n(d_i)$</th>
<th>$n(1 \text{ mm})$</th>
<th>$\phi(1 \text{ mm})$</th>
<th>$\psi(1 \text{ mm})$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simple</td>
<td>2442390</td>
<td>475640</td>
<td>208944</td>
<td>27799</td>
<td>5.84%</td>
<td>13.3%</td>
</tr>
<tr>
<td>Adv.</td>
<td>2442390</td>
<td>475640</td>
<td>137372</td>
<td>432</td>
<td>0.09%</td>
<td>0.3%</td>
</tr>
<tr>
<td>Exp.</td>
<td>Unknown</td>
<td>Unknown</td>
<td>500</td>
<td>1</td>
<td>Unknown</td>
<td>0.2%</td>
</tr>
</tbody>
</table>

3.6 Conclusions

The flow of molten steel, and the transport and capture of argon gas bubbles have been simulated and compared with plant measurements in a continuous slab caster. A two-way coupled Eulerian-Lagrangian model combined with a bubble tracking DRW model has been applied to predict asymmetrical flow on the top surface, which were seen in nailboard measurements. Two capture criteria were implemented, and the advanced capture criterion showed better agreement with measurements of the number, locations, and sizes of captured bubbles, especially for larger bubbles. The relative capture fraction of large (1 mm) bubbles of 0.3% was close to the measured 0.2%, and occurred very near the top surface. Some important findings are summarized below:

1. The slide gate does not cause biased cross-flow at the top surface for single phase flow. However, when Ar gas is injected with the sliding gate moving towards the IR, more Ar bubbles leave the ports towards the IR and lead to cross flow on the top surface from IR to OR, and corresponding non-uniform bubble capture on the IR and OR;

2. Almost 85% of small ($<0.08 \text{ mm}$) bubbles are captured. A very small fraction of large bubbles is captured ($<0.02\%$). This fraction drops almost one order of magnitude with increasing bubble size from 1 to 2 mm and from 2 to 3 mm;

3. The removal fraction of large bubbles ($1 \leq d_p \leq 5 \text{ mm}$) was 99.98%, for medium bubbles ($0.1 \leq d_p \leq 0.3 \text{ mm}$) was around 48.5% and for small bubbles ($d_p < 0.1 \text{ mm}$) was close to 15%;

4. Most bubbles are captured very near to the meniscus. Deeper than 0.5 m below the meniscus, the capture fraction of medium bubbles is about half that of small bubbles;

5. The predicted bubbles captured shows similar trends as the plant measurements, except in the first layer near to the outer surfaces, where more bubbles are captured than predicted. This might be due to the model neglecting the effects of meniscus hook capture, or assuming immediate escape of bubbles that reach the top surface.

3.7 References


Chapter 4

Effects of External Magnetic Field on Shear Driven Flow

To investigate the effects of EMBr on the molten steel flow in the mold region of the caster and accelerate the solution procedure, a magnetohydrodynamics (MHD) module was implemented into CUFLOW. To validate the MHD solver and test the performance, the code is used to solve the flow in a lid-driven cavity with external magnetic field and the results are compared published results. Then the code is used to study the flow in a shear driven cavity, which has similar structure as the top roll in the mold region of a continuous caster of steel. Comparing with the real top roll in the caster, which is driven by an unsteady jet, this simplified problem setup uses a moving wall to drive the conducting liquid in the cavity. The physics in both cases are similar: shear flow with a big vortex. The fundamental study on this simplified geometry provides insight of the effects of Lorentz force on the motion of conducting liquid, which is beneficial to many other science and engineering fields. The work presented in this chapter has been published in the Journal of Fluids Engineering.[1].

4.1 Introduction

The flow in a cavity driven by the shear induced by motion of one of the walls has been a classical problem in fluid mechanics and numerous research papers have appeared on the characterization of the vortices generated in such a flow [2–42]. The widely studied case is the flow generated by the motion of the top wall of a square cavity in which a primary recirculating eddy and two smaller corner eddies at the bottom left and bottom right corners are generated. Bench-mark solutions of two-dimensional Newtonian flow in a square cavity have been published by several researchers e.g. [2–11]. A number of studies for three-dimensional flow in a cube driven by the top wall have also been published e.g. [18–42], illustrating the rich three-dimensional flow structures that are generated due to presence of the walls in the spanwise direction and by the instabilities of the curved streamlines. It is seen that on the bottom wall, streamwise vortical structures akin to Taylor-Göertler-like (TGL) vortices are generated after a critical Reynolds number is exceeded. In addition, the flow becomes progressively more complex, eventually becoming unsteady and turbulent [37–39].
The rectangular shapes have been the most widely studied geometries, and have become standard problems for development and validation of computational algorithms and codes. A smaller number of studies have been performed on cavities of other shapes, such as trapezoidal [43–45], triangular [43, 46, 47], semi-circular [48, 49] and other complex shapes [50].

When a magnetic field is applied to an electrically conducting moving fluid, an electric current field is generated. This current field, together with the magnetic field, generates a force field \( \vec{F} = \vec{J} \times \vec{B} \) where \( \vec{J} \) and \( \vec{B} \) are electrical current and magnetic field vectors, respectively) that brakes the fluid motion. The modified velocity field in turn changes the current field and the electromagnetic force on fluid. This two-way interaction has profound effects on transport phenomena in various industries such as steel making, crystal growth, welding, aluminum smelting, fusion power generation, Magnetohydrodynamics (MHD) coal fired power plants, etc. Such MHD forces have also been known to modify the turbulence structure including completely laminarizing a turbulent flow at sufficiently large magnetic fields and interaction parameters.

The effect of a magnetic field on a driven cavity flow is quite interesting because of the complex vortical structures that interact in a complex way with the magnetic field. To our knowledge, only a limited number of studies have examined such flows, primarily in two dimensions. Al-Salem et al. [51] used a finite volume method to investigate the effect of a magnetic field on heat transfer in a lid-driven cavity with bottom wall subject to a linear temperature distribution. They found that increasing the Hartmann number decreases the strength of the flow and therefore the heat transfer rate is also reduced. Their study also showed that due to the non-uniform temperature distribution along the bottom wall, the heat transfer rate can be improved if the lid moves in the same direction as the wall temperature decreases. Oztop et al. [52] studied the effect of a magnetic field on heat transfer in a two dimensional lid-driven square cavity with a corner heater for several different Hartmann numbers and Grashof numbers at a fixed Reynolds number of \( Re = 100 \). The lid was moved in the positive coordinate direction while the magnetic field was applied in the negative coordinate direction. They found that a increase in Hartmann number causes the top eddy to shrink and become closer to the moving lid, and simultaneously the bottom eddy becomes larger. The strength of the fluid circulation was reduced and the thickness of the thermal boundary layer was increased as the Hartmann number was increased. The local Nusselt number along the bottom wall consequently decreased with Hartmann number.

Sivasankaran et al. [53] also used a finite volume code to investigate the mixed convection in a square lid driven cavity with sinusoidal boundary temperature at the vertical walls in the presence of an external magnetic field for a fixed Reynolds number of 100. Results for three different Hartmann numbers (0, 25 and 100) are reported and they found that increasing the Hartmann number restricts the flow circulation in the region close to the top lid. Their results also showed that if forced convection dominates the flow...
behavior, increasing the Hartmann number does not affect the heat transfer at the side walls. However, if the flow is in the regime of buoyancy-driven convection, increasing the Hartmann number leads to decrease of total heat transfer rate. Yu et al. [54] studied the effect of Hartmann number on natural convection heat transfer in a 2D square cavity at different Rayleigh numbers using a streamfunction-vorticity scheme. In their study, the vertical walls of the cavity were maintained at different temperatures. For $Ha = 100$ and $Ra = 10^4$, they investigated the effect of inclination angle of the magnetic field. Their results showed that for large Hartmann numbers, the flow structure is highly influenced by the inclination angle of the magnetic field, but the temperature field is not sensitive to the inclination angle if the magnetic field is dominant.

In all the above studies [51–54] the Maxwell’s equations for the electrical potential were not solved and the effect of Lorentz force on the momentum equations was included by adding a source term which is linearly proportional to the local velocity and square of the external magnetic field (i.e. $F = vB^2\sigma/\rho$).

Shatrov et al. [55] reported results of a 2D square lid-driven cavity flow and linear stability analysis (LSA) of a 3D lid-driven cavity flow problem with external magnetic field applied in the same direction as the moving lid. For the 2D MHD problem, they found that the strength of the eddy is weakened by the external magnetic field. With increasing magnetic field the primary eddy changes from a circular shape to an elliptical shape. With further increase in the strength of the magnetic field, the secondary eddy begins to occupy the bottom part (from left to right) of the square cavity. Their linear stability analysis showed that for the 3D lid-driven cavity the instability of the flow can be dampened by increasing the strength of external magnetic field. For $Re = 3100$, there exist several branches of the neutral stability curve and an increase of the magnetic field strength may lead to a transition from a stable flow to an oscillatory flow.

The three-dimensional flow of a Newtonian fluid in a cube driven by the top wall without a magnetic field has also been the subject a large number of numerical studies. Interest in such studies dates back to 1975 [18], and has been maintained steadily for the past 40 years (e.g. [18–41]). There have been a variety of techniques used to compute essentially the same flow, albeit with different accuracy, grid, and Reynolds numbers. Finite volume methods [19, 20, 22, 27], finite-element methods [29, 31], spectral methods [21, 30, 33], velocity vorticity methods [23], and Lattice Boltzmann methods [34, 37, 38] have been used. The primary features observed are nearly the same in all these studies. At low Reynolds numbers, the flow is similar to the flow in a two-dimensional cavity, except for the boundary layers on the spanwise walls (if the spanwise direction is periodic, exactly two-dimensional solutions are obtained). The central plane of the three-dimensional domain consists of a primary eddy and two secondary eddies in the right and left bottom corners. However, the primary feature of three-dimensional analyses is capturing possible three-dimensional structures even in a nominally two-dimensional (i.e. periodic) geometry. For example, at reasonably high
Reynolds numbers, it has been observed that the recirculating primary eddy generates centrifugal instabilities [30], and Taylor-Görtler type vortices are formed near the bottom walls [31]. Freitas et al. [19] have applied a finite volume code REBUFFS to study the lid-driven 3D cavity flow at Re=3200 and compared their results with their own experiments. They used a $32 \times 32 \times 45$ grid and the SIMPLE [56] algorithm with QUICK [57] scheme and solved only half of the cavity assuming the flow to be symmetric about the middle plane. They were able to capture the TGL vortices also found in the experiments. The size and the location of those vortices were observed to be time dependent. Mean velocities on the central lines of the symmetry plane were satisfactorily compared with laser-Doppler velocity measurements at $Re = 3300$.

Based on the above literature study, we conclude that to the best of our knowledge, there have been no published studies which have systematically presented the effect of a magnetic field on an electrical conducting fluid in a 3D lid driven cavity at different Reynolds numbers and for several magnetic field strengths. In the presented work, we have conducted several highly resolved simulations of such flows. Three dimensional MHD flows are obviously more computationally expensive over two-dimensional simulations. However, with the development of parallel computers, it is now possible to perform large scale three-dimensional flow calculations in reasonable computing times. The Graphical Processing Unit (GPU) is a highly parallel computing platform available on a desktop. The GPU is a highly parallel, multi-threaded, many core processor with a large computational power and high memory bandwidth. In recent years, it has been increasingly recognized that the data parallel features of a GPU can be effectively exploited to perform large scale scientific computations to achieve greater speeds without adding to the cost of performing the computation on a CPU. Recently a considerable amount of interest has been generated on the use of the GPU as a data parallel computing platform [58, 59]. A recent review of implementation of Computational Fluid Dynamics (CFD) codes on GPUs has been given in [58], which refers to a large numbers of papers. Several researchers have developed/ported CFD software to GPUs and found significant speed-ups (10 to 50 times depending on algorithm, approach and implementation) over a single core CPU. The GPU is a highly parallel, multi-threaded, many core processor with a large computational power and high memory bandwidth.

In this paper, we study the motion of an electrically conducting fluid in a cavity with a moving wall and an external magnetic field. The 3D Navier-Stokes equations are solved using the Harlow-Welch[58] algorithm. A considerable speedup is obtained by implementing the algorithm on the GPU. Streamline patterns and velocity profiles at the center lines for different Reynolds numbers and magnetic field strengths are presented. This paper is organized by first describing the governing equations and solution procedure in Section 2. The validation and grid independency studies are presented in Section 3. Section 4 discusses results of a number of well-resolved calculations. Finally, the important results are summarized.
4.2 Governing Equations and Solution Procedure

The problem being solved is the flow of an incompressible electrically conducting fluid in an electrically insulated three dimensional cavity with the top wall moving. The magnetic field is applied in the same direction as the moving wall. In this work, the ratio of the induced magnetic field to the external magnetic field is assumed to be much less than one, and therefore the induced magnetic field generated by electromagnetic induction is neglected. The set of governing equations for this flow is given by Eqn. (4.1) - (4.6). The electrical current density $\vec{J}$ can be computed through the Ohm’s law as given by Eqn. (4.1), and for a well conducting material the current conservation law can be written as Eqn. (4.2). Therefore, the electric potential $\Phi$ satisfies Eqn. (4.3). The Lorentz force $\vec{F}$ is the cross product of current density and external magnetic field $B_0$, as given in Eqn. (4.4). The continuity and momentum equations are given by Eqn. (4.5) and (4.6), respectively. In the dimensionless equations, distances were non-dimensionalized with the height of the cavity $L$, velocities were non-dimensionalized using the moving wall speed $U$, pressure was non-dimensionalized with $\rho U^2$ and time was non-dimensionalized with $L/U$.

\[
\vec{J} = \sigma \left( -\nabla \Phi + \vec{u} \times \vec{B}_0 \right) \quad (4.1)
\]

\[
\nabla \cdot \vec{J} = 0 \quad (4.2)
\]

\[
\nabla^2 \Phi = \nabla \cdot \left( \vec{u} \times \vec{B}_0 \right) \quad (4.3)
\]

\[
\vec{F} = \vec{J} \times \vec{B}_0 \quad (4.4)
\]

\[
\nabla \cdot \vec{u} = 0 \quad (4.5)
\]

\[
\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla) \vec{u} = -\nabla p + \frac{1}{Re} \nabla^2 \vec{u} + N \vec{F} \quad (4.6)
\]

Here, $\vec{u} = (u, v, w)$ is the velocity vector, $p$ is the dimensionless pressure, $t$ is dimensionless time; $Re$ is Reynolds number; $N$ is Stuart number (also known as magnetic interaction parameter) that describes the
ratio of electromagnetic to inertial forces and $N = Ha^2 / Re$, where the Hartmann number $Ha$ is defined as $Ha = B_0 L (\sigma/\mu)^{1/2}$; $\sigma$ and $\mu$ are electrical conductivity and dynamic viscosity of the fluid, respectively. $B_0$ is strength of the applied external magnetic field. In this work, we have conducted simulations for Reynolds number $Re$ from 400 to 5000 and the Stuart number $N$ from 0.0 to 2.0. The boundary conditions at the walls are no-penetration and no-slip for momentum and insulated for the current flow.

The above equations are solved with an in-house code, CUFLOW [60–62]. CUFLOW is a general purpose code for simulating laminar and turbulent flows in complex domains. The code employs Cartesian grids to integrate the three-dimensional unsteady incompressible Navier-Stokes equations. The continuity and momentum equations are solved using a fractional step method. Brief details of the solution algorithm are provided below and complete details are available in references [61, 62].

In the first step of the fractional step method, intermediate velocities are determined by solving the momentum equations without the pressure gradient terms. The discretized equations are derived by a finite-volume framework using central differencing for both convection and diffusion terms on a collocated grid. For the temporal differencing, the second-order accurate Adams-Bashforth scheme is used. The discretized equations in the absence of pressure gradient terms are given by:

$$\rho \left( \frac{\hat{u}_i - u_i^n}{\Delta t} \right) = \frac{3}{2} H_{u_i}^n - \frac{1}{2} H_{u_i}^{n-1}$$

(4.7)

Here $u_i$ denotes the velocity in the i direction, and $u_1$ stands for $u$ velocity, $u_2$ stands for $v$ velocity and $u_3$ stands for $w$ velocity. In the second step, the continuity equation is transformed to a pressure-Poisson equation given by:

$$\frac{\rho}{\Delta t} \frac{\partial \hat{u}_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\partial p}{\partial x_i} \right)^{n+1}$$

(4.8)

Equation (4.8) for pressure is solved efficiently by a V-cycle multigrid method, and red-black SOR (with over-relaxation parameter of 1.6). After computing the pressure at $n + 1$ time step from Eqn. (4.8), the velocity components are updated for the effects of the pressure gradient term. For steady state calculations, the algorithm is marched in time to desired degree of convergence in time.

The above algorithm has been programmed to run entirely on a GPU. The GPU has been observed to provide a factor upwards of twenty speed up over a single core CPU. GPU programming can be done in several different languages such as OpenCL, CUDA, OpenGL, etc. Of these, OpenCL and CUDA are most commonly used. We have implemented the algorithm in CUDA Fortran. CUDA Fortran is supported by the Portland Group (PGI) Fortran compiler. The grid generation, initial conditions, and boundary conditions are first created on CPU and data are then copied to the GPU. For each computational step, a separate
GPU kernel is launched. The flow fields from the GPU are copied periodically to the CPU for plotting and interrogation. In the current version of CUFLOW, an unstructured one-dimensional data structure is employed in order to simulate complex geometries. The GPU launches data in blocks of pre-specified sizes and each block is assigned to one streaming multiprocessor (SMP). The SMP launches threads which are then assigned to kernels. A kernel is a set of instructions assigned to one thread to be executed independent of other data. Thus, GPU is a data parallel computer, operating same instructions in parallel on multiple data. As mentioned earlier, CUFLOW uses an explicit algorithm for momentum equations and a red-black Successive Over Relaxation (SOR) algorithm for the pressure-Poisson equation. Both these are data parallel algorithms and easily map to a GPU. CUFLOW has been successfully used to perform several Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) of turbulence such as in circular and triangular ducts, the effects of a micro ramp on the film cooling effectiveness [62], and flow and heat transfer in the mold region of continuous casters of steel[61]. Details of these results with CUFLOW are given in [60–62].

4.3 Code Validation and Grid Independent Study

Previously, CUFLOW was validated in a number of flow problems, including turbulent flow in square ducts [50], MHD flow in channels and ducts [59], continuous casting of steel [61, 63], and in film cooling flow with vortex generators [62]. However, the current version of CUFLOW was modified to use a collocated grid (versus a staggered grid in the previous version), and hence was validated again for flow in a driven cube with a Newtonian fluid against previous simulations of Ku et al. [21]. Figure 4.1 shows the comparison of the velocity profiles at mid-span along horizontal and vertical lines. The solution of Ku et al. [21] was obtained with a spectral technique and our simulations were performed with a $128 \times 128 \times 128$ uniform finite volume grid. Several coarser grids were also computed, and the $128 \times 128 \times 128$ was able to match very well with the results from Ku et al. [21]. Since the code used a GPU, the computations were quite fast and did not require much GPU time. The calculations were performed by marching in time to steady state, and a steady state was judged by successive changes of velocities to be less than $1.0 \times 10^{-5}$ for a nominal value of 1.0.

The validation of the MHD part of the solver was done by solving a lid-driven cavity flow in a cavity of aspect ratio $x : y : z = 1 : 1 : 8$ with a grid of $128 \times 128 \times 512$ finite volumes for CUFLOW and $128 \times 128 \times 100$ for ANSYS Fluent [64]. The magnetic field was applied in the $x$ direction and top wall ($y = 1$) was moving in the $x$ direction with velocity of 1.0. The Reynolds number of this problem was 5000 and the Stuart number $N = 5$. A 2D version of this problem, which assumes infinite length in $z$ direction, has been previously
reported by Shatrov et al. [55]. In Fig. 4.2 the u velocities on vertical centerline of the symmetry plane of the 3D simulations are compared with the velocity from the 2D simulation. The agreement between the two is excellent, thus validating the solution of the MHD equations as well. Next, grid independency studies were carried out to establish an adequate grid that gives accurate results.

Calculations were made with $64 \times 64 \times 64$, $128 \times 128 \times 128$ and $192 \times 192 \times 192$ finite volumes for $Re = 3200$ and $N = 0.25$. The u velocity on vertical centerline in the symmetry plane ($z = 0.5$) and v velocity on the horizontal centerline are plotted in Fig. 4.3.

These comparisons show that results from $64 \times 64 \times 64$ grid are not accurate enough, but results with $128 \times 128 \times 128$ and $192 \times 192 \times 192$ grids are very close to each other. A little mismatch at the maximum v velocity is seen between the $128 \times 128 \times 128$ and the $192 \times 192 \times 192$ grid. Therefore, in the current study for Reynolds numbers 3200 and less the $128 \times 128 \times 128$ grid is used whereas the $192 \times 192 \times 192$ grid is used for Re of 5000.
Figure 4.2: Validation result of 2D cavity MHD flow at $Re = 5000$ and $N = 5$.

Figure 4.3: Grid independency study, $Re = 3200$ and $N = 0.25$ in symmetry plane ($z=0.5$)
4.4 Results and Discussions

In this work, a total of 23 simulations were performed with Reynolds number $Re$ ranging from 400 to 5000 and Stuart number $N$ ranging from 0.0 to 2.0. Tab. 4.1 lists the parameters for these computations. The top lid, represented by surface $y = 1$, moves in the $x$ direction with the magnetic field applied in the same direction. Figure 4.4 shows the geometrical configuration.

![Figure 4.4: The lid driven cavity with an external magnetic field](image)

4.4.1 Simulations at $Re = 400$

Four simulations with different Stuart numbers have been conducted for $Re = 400$. Due to the low Reynolds number, steady state solutions were obtained for all these cases. Figure 4.5 shows the contours of velocity magnitude in different constant $z$ planes. The results show that without the magnetic field the primary eddy occupies almost the entire cavity and the velocity contours show that the streamwise behavior at $z = 0.3$ is very similar to that $z = 0.5$ which is the symmetry plane. Figure 4.6 shows contours of velocity magnitude in different $z$ planes at $N = 1.0$. Comparing these plots with those for $N = 0$, we observe that the main eddy is compressed significantly with the region of velocity magnitude greater than 0.1 (10% of top wall velocity) occupying the top half of the cavity. Increasing $N$ further to 2.0, the recirculating region moves up further and takes around top 40% of the cavity, as shown in Fig. 4.7.

The streamlines on the middle $z$ plane ($z = 0.5$) are shown in Fig. 4.8. Figure 4.8a shows that with no magnetic field, there is a relatively small secondary eddy at the bottom right corner of the cavity. However, as $N$ increases, the main eddy moves up and this secondary eddy grows. Eventually when $N$ exceeds 0.25
<table>
<thead>
<tr>
<th>No.</th>
<th>Re</th>
<th>N</th>
<th>Grid</th>
<th>Steady/Unsteady</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>400</td>
<td>0</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>2</td>
<td>400</td>
<td>0.25</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>3</td>
<td>400</td>
<td>0.5</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>4</td>
<td>400</td>
<td>1</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>5</td>
<td>400</td>
<td>2</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>6</td>
<td>2000</td>
<td>0</td>
<td>128</td>
<td>Unsteady</td>
</tr>
<tr>
<td>7</td>
<td>2000</td>
<td>0.0125</td>
<td>128</td>
<td>Unsteady</td>
</tr>
<tr>
<td>8</td>
<td>2000</td>
<td>0.05</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>9</td>
<td>2000</td>
<td>0.25</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>10</td>
<td>2000</td>
<td>0.5</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>11</td>
<td>2000</td>
<td>1</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>12</td>
<td>2000</td>
<td>2</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>13</td>
<td>3200</td>
<td>0</td>
<td>128</td>
<td>Unsteady</td>
</tr>
<tr>
<td>14</td>
<td>3200</td>
<td>0.04</td>
<td>128</td>
<td>Unsteady</td>
</tr>
<tr>
<td>15</td>
<td>3200</td>
<td>0.0625</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>16</td>
<td>3200</td>
<td>0.09</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>17</td>
<td>3200</td>
<td>0.25</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>18</td>
<td>3200</td>
<td>0.5</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>19</td>
<td>3200</td>
<td>1</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>20</td>
<td>3200</td>
<td>2</td>
<td>128</td>
<td>Steady</td>
</tr>
<tr>
<td>21</td>
<td>5000</td>
<td>0</td>
<td>192</td>
<td>Unsteady</td>
</tr>
<tr>
<td>22</td>
<td>5000</td>
<td>0.25</td>
<td>192</td>
<td>Steady</td>
</tr>
<tr>
<td>23</td>
<td>5000</td>
<td>0.5</td>
<td>192</td>
<td>Steady</td>
</tr>
</tbody>
</table>

The secondary eddy occupies completely the bottom half of the cavity. However, the velocity in the bottom region of the cavity is very small and the flow is almost stationary for the Stuart number $N \geq 1$. Figure 4.8d shows the streamlines for case $Re = 400$ and $N = 1.00$, and the result shows that in this case the bottom eddy has about equal size as the top main eddy. However, the eye of the bottom eddy is at the center of the bottom half of the cavity, while the eye of the top eddy is close to the downstream corner. It is important to mention that the strength of the secondary eddy is still much smaller than that of the primary eddy. This can also be seen from Fig. 4.10a where the $u$ velocity is very small for the region $y < 0.3$.

The $u$, $v$ and $w$ velocities on the line passing through the center and between the two side walls are shown in Fig. 4.9. When no magnetic field is applied, the $u$ velocity shows a "W-like" shape and $u$ has a negative through the line. As $N$ increases, the main eddy moves up along with the bottom part of the eddy also moving up. This makes the $u$ velocity more negative. With increasing $N$, this velocity increases (decreasing in magnitude) with even a change in sign near the side walls for $N = 2$. The $v$ velocity along the spanwise centerline is positive for all the different Stuart numbers. When no magnetic field is applied the peak $v$ velocity on the spanwise centerline is seen at region about 0.1 away from the side walls, and has a local peak $v$ velocity at the middle with $z = 0.5$. The results show that as $N$ increases, the $v$ velocity becomes smaller and the profile is flatter. The $w$ velocity on the spanwise centerline is plotted in Fig. 4.9(c).
Figure 4.5: Velocity magnitude contour at Re=400 and N=0.0 on (a) z=0.05 (b) z=0.1 (c) z=0.3 and (d) z=0.5 which shows a wavy shape for low values of N. For large N, the $w$ velocity is seen to diminish to nearly a zero value. The peak $w$ velocity happens in the region around $z = 0.2$ away for the case when no magnetic field is applied, and varies linearly between $z = 0.2$ to $z = 0.8$. The results also show that as N increases, the magnitude of $w$ velocity decreases.

Figure 4.10 shows the $u$ velocity on the vertical centerline and $v$ velocity on the horizontal centerline in the symmetry plane ($z = 0.5$). Selected points on these lines are tabulated in Tab. 4.2 and Tab. 4.3. Figure 4.10a shows that as $N$ increases, the negative peak of $u$ velocity shifts upwards toward the top moving wall. It also shows that the maximum magnitude of the negative $u$ velocity is only slightly affected by increasing $N$, so applying the magnetic field only shifts the negative peak $u$ velocity from bottom of the cavity to a region closer to the top moving wall. It also shows with $N \geq 1$, the bottom half of the cavity is almost stationary and the primary eddy is confined in the top half of the cavity. Figure 4.10b shows the $v$ velocity on horizontal centerline. It is seen that the peak of $v$ velocity decreases with increasing $N$, and as $N$ increases from 0 to 1, the negative peak value is reduced from $-0.38$ to $-0.016$. This is a result of the suppression of flow circulation and shifting of the center of flow upwards due to the magnetic field. Because the magnetic field is applied in the $x$ direction, and the damping effect is mainly in the $y$ direction.
Figure 4.6: Velocity magnitude contour at Re=400 and N=1.0 on (a) z=0.05 (b) z=0.1 (c) z=0.2 and (d) z=0.5

Figure 4.7: Velocity magnitude contour at Re=400 and N=2.0 on (a) z=0.05 (b) z=0.1 (c) z=0.2 and (d) z=0.5

Table 4.2: $Re = 400$ $u$ velocity on vertical center line $x = 0.5$

<table>
<thead>
<tr>
<th>$y$</th>
<th>N=0.00</th>
<th>N=0.25</th>
<th>N=0.50</th>
<th>N=1.00</th>
<th>N=2.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.10</td>
<td>$-1.43 \times 10^{-1}$</td>
<td>$-8.16 \times 10^{-4}$</td>
<td>$1.06 \times 10^{-2}$</td>
<td>$1.00 \times 10^{-2}$</td>
<td>$4.86 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.20</td>
<td>$-2.22 \times 10^{-1}$</td>
<td>$-8.38 \times 10^{-2}$</td>
<td>$-1.30 \times 10^{-3}$</td>
<td>$9.94 \times 10^{-3}$</td>
<td>$7.15 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.30</td>
<td>$-2.21 \times 10^{-1}$</td>
<td>$-1.79 \times 10^{-1}$</td>
<td>$-4.01 \times 10^{-2}$</td>
<td>$-7.73 \times 10^{-4}$</td>
<td>$8.08 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.40</td>
<td>$-1.40 \times 10^{-1}$</td>
<td>$-1.02 \times 10^{-1}$</td>
<td>$-1.08 \times 10^{-1}$</td>
<td>$-3.12 \times 10^{-2}$</td>
<td>$1.04 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.50</td>
<td>$-5.02 \times 10^{-2}$</td>
<td>$-1.13 \times 10^{-2}$</td>
<td>$-1.80 \times 10^{-1}$</td>
<td>$-9.28 \times 10^{-2}$</td>
<td>$-3.02 \times 10^{-2}$</td>
</tr>
<tr>
<td>0.60</td>
<td>$1.20 \times 10^{-2}$</td>
<td>$2.12 \times 10^{-2}$</td>
<td>$-1.91 \times 10^{-1}$</td>
<td>$-1.65 \times 10^{-1}$</td>
<td>$-9.58 \times 10^{-2}$</td>
</tr>
<tr>
<td>0.70</td>
<td>$6.20 \times 10^{-2}$</td>
<td>$3.18 \times 10^{-2}$</td>
<td>$-1.19 \times 10^{-1}$</td>
<td>$-1.90 \times 10^{-1}$</td>
<td>$-1.85 \times 10^{-1}$</td>
</tr>
<tr>
<td>0.80</td>
<td>$1.10 \times 10^{-1}$</td>
<td>$3.19 \times 10^{-2}$</td>
<td>$-2.24 \times 10^{-2}$</td>
<td>$-1.17 \times 10^{-1}$</td>
<td>$-1.90 \times 10^{-1}$</td>
</tr>
<tr>
<td>0.90</td>
<td>$2.25 \times 10^{-1}$</td>
<td>$3.56 \times 10^{-2}$</td>
<td>$1.34 \times 10^{-1}$</td>
<td>$7.61 \times 10^{-2}$</td>
<td>$1.94 \times 10^{-2}$</td>
</tr>
</tbody>
</table>
Figure 4.8: Streamlines in $z = 0.5$ plane, $Re = 400$ for different $N$
Figure 4.9: (a) $u$, (b) $v$ and (c) $w$ velocities on spanwise centerline.

Figure 4.10: Centerline velocities in $z = 0.5$ plane, $Re = 400$ for different magnetic fields.
Table 4.3: \(Re = 400\) \(v\) velocities on horizontal line \(y = 0.5\)

<table>
<thead>
<tr>
<th>(x)</th>
<th>(N=0.00)</th>
<th>(N=0.25)</th>
<th>(N=0.50)</th>
<th>(N=1.00)</th>
<th>(N=2.00)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.10</td>
<td>(1.91 \times 10^{-1})</td>
<td>(6.00 \times 10^{-2})</td>
<td>(9.71 \times 10^{-3})</td>
<td>(-9.60 \times 10^{-3})</td>
<td>(-1.52 \times 10^{-2})</td>
</tr>
<tr>
<td>0.20</td>
<td>(1.98 \times 10^{-1})</td>
<td>(8.21 \times 10^{-2})</td>
<td>(2.36 \times 10^{-2})</td>
<td>(-5.71 \times 10^{-3})</td>
<td>(-1.32 \times 10^{-2})</td>
</tr>
<tr>
<td>0.30</td>
<td>(1.62 \times 10^{-1})</td>
<td>(9.04 \times 10^{-2})</td>
<td>(3.71 \times 10^{-2})</td>
<td>(1.23 \times 10^{-3})</td>
<td>(-8.69 \times 10^{-3})</td>
</tr>
<tr>
<td>0.40</td>
<td>(1.16 \times 10^{-1})</td>
<td>(8.22 \times 10^{-2})</td>
<td>(4.25 \times 10^{-2})</td>
<td>(7.79 \times 10^{-3})</td>
<td>(-3.50 \times 10^{-3})</td>
</tr>
<tr>
<td>0.50</td>
<td>(6.04 \times 10^{-2})</td>
<td>(5.18 \times 10^{-2})</td>
<td>(3.32 \times 10^{-2})</td>
<td>(1.09 \times 10^{-2})</td>
<td>(1.53 \times 10^{-3})</td>
</tr>
<tr>
<td>0.60</td>
<td>(-6.45 \times 10^{-3})</td>
<td>(-5.70 \times 10^{-3})</td>
<td>(3.51 \times 10^{-3})</td>
<td>(7.44 \times 10^{-3})</td>
<td>(5.44 \times 10^{-3})</td>
</tr>
<tr>
<td>0.70</td>
<td>(-1.22 \times 10^{-1})</td>
<td>(-1.08 \times 10^{-1})</td>
<td>(-5.23 \times 10^{-2})</td>
<td>(-4.69 \times 10^{-3})</td>
<td>(8.31 \times 10^{-3})</td>
</tr>
<tr>
<td>0.80</td>
<td>(-3.18 \times 10^{-1})</td>
<td>(-2.05 \times 10^{-1})</td>
<td>(-9.43 \times 10^{-2})</td>
<td>(-1.60 \times 10^{-2})</td>
<td>(9.82 \times 10^{-3})</td>
</tr>
<tr>
<td>0.90</td>
<td>(-3.26 \times 10^{-1})</td>
<td>(-1.43 \times 10^{-1})</td>
<td>(-6.08 \times 10^{-2})</td>
<td>(-1.08 \times 10^{-2})</td>
<td>(1.06 \times 10^{-2})</td>
</tr>
</tbody>
</table>
4.4.2 Simulations at Re = 2000

Increasing the Reynolds number increases the nonlinearity and strength of the advection terms. The Re = 2000 simulations were also performed with a 128\(^3\) grid. Figure 4.11 shows the streamlines on the symmetry plane (\(z = 0.5\)) for several values of \(N\). We observe that when there is no magnetic field or when only a weak magnetic field (\(N = 0.0125\)) is applied, the flow is unsteady. Therefore the lines in the plots are instantaneous streaklines obtained by releasing massless tracing particles on a typical frozen instantaneous flow field. For \(N = 0\), Fig. 4.11 shows that without the magnetic field, there are two small secondary eddies located at the bottom corners. After applying a modest magnetic field, with \(N = 0.05\) there are still two secondary eddies at the bottom corners, but the flow is steady for this Stuart number and higher. Comparing the six plots shown in Fig. 4.11, we see that with increasing \(N\), the main eddy moves upward and toward the top downstream corner of the cavity. This trend is similar to the 2D cases described in \([52, 53]\). Comparing the shape of the primary eddy for different values of \(N\), it is seen that the shape of the primary eddy also changes from a circular shape to an elliptic shape similar to the 2D case reported by \([55]\).

Figure 4.11c shows the streamlines in the central \(z\) plane for \(Re = 2000\) and \(N = 0.25\). In this case, also the flow is steady with two secondary eddies still seen at the bottom of the cavity. However, there is a trend toward the two secondary eddies to interact with each other and merge into a bigger one. This is seen with increasing \(N\) further to 0.5. Figure 4.11d for \(N = 0.5\) shows that the two bottom secondary eddies form a large eddy that occupies the bottom of the cavity. As \(N\) is further increased to 1.0, the top main eddy shrinks and bottom eddy grows even larger than the primary eddy. But the strength of the bottom eddy is much smaller than the primary one. Eventually when \(N = 2.0\) the main eddy shrinks to the very top of the cavity and only occupies top \(1/3\) of the cavity as shown in Fig. 4.11f. In this case, secondary and weak tertiary eddies start to appear in the bottom part of the cavity.

The \(u\) velocity on vertical centerlines and \(v\) velocity on horizontal centerline for \(z = 0.5\) are shown in Fig. 4.12. For different \(N\), Fig. 4.12a illustrates that with increasing \(N\) the peak of \(u\) is moving away from the bottom wall and the bottom wall boundary layer thickness is growing. As the peak moving away from the wall, the peak value remains almost the same and is not affected considerably. Figure 4.12a shows that when \(N = 2.00\), the flow field below \(y = 0.5\) is almost stationary. The main effect of increasing \(N\) on \(u\) is the shift in the peak \(u\) velocity towards the moving lid. Figure 4.12b illustrates that increasing \(N\), the peak \(v\) velocity is reduced and the boundary layer that adjacent to the sidewall becomes thicker. The growth of the boundary layer pushes the peak \(v\) velocity away from the side walls. Selected points in Fig. 4.12a and 4.12b are tabulated in Tab. 4.4 and 4.5, respectively.

Figure 4.13 shows velocity vectors in the middle \(x\) plane (\(x = 0.5\)). As seen in Fig. 4.13a, when there is
no magnetic field, four vortices (Taylor-Görtler-like vortex pairs) are seen at the four corners, and the two vortices at the bottom are much stronger than those in the upper region. However, for $N = 1.0$ these four vortices disappear as a results of the damping effect of the magnetic field.
Figure 4.11: Streamlines in \( z = 0.5 \) plane, \( Re = 2000 \) for different \( N \)
(a) $u$ velocity along vertical centerline

(b) $v$ velocity along horizontal centerline

Figure 4.12: Centerline velocity in $z = 0.5$ plane, $Re = 2000$

(a) Velocity vector in middle $x$ plane, $N = 0.00$

(b) Velocity vector in middle $x$ plane, $N = 1.00$

Figure 4.13: Velocity vector in middle $x$ plane for $Re = 2000$
4.4.3 Simulations at Re = 3200

Further increase of Re to 3200 makes the flow more nonlinear and the flow is seen to be unsteady for N = 0 and N = 0.04. Figure ?? and ?? show the instantaneous streakline plot of the flow fields generated with N = 0 and N = 0.04. Increase of N to 0.0625 makes the flow steady, as shown in Fig. ?? . It is seen that for N = 0.04 and N = 0.0625, the eye of the primary vortex moves down below the position seen for N = 0. At N = 0.0625, two secondary eddies are formed at the bottom corner of the cavity. Similar streamline pattern is seen for N = 0.09. Further increase of the magnetic interaction parameter shrinks the top circulation eddy and modifies the shape of the eddy from a circular to elliptical, as seen also for Re = 2000. The general flow patterns observed until N = 2.0 are similar to those seen at Re = 2000. The bottom two secondary eddies seen at N = 0.25 merge into a single large secondary eddy in Fig. ?? at N = 0.5. At N = 1.0 and 2.0, the primary eddy shrinks and only occupies the top 1/3 of the cavity. Below the primary eddy one secondary and one tertiary eddy show up. However these two eddies are much weaker that the primary one.

Figure 4.15 shows the $u$ velocity on vertical centerline and $v$ velocity on horizontal centerline on the symmetry plane for Re = 3200, and again selected points are tabulated in Tab. 4.6 and 4.7, respectively. As seen before, Fig. 4.15a shows that the location of the peak $u$ velocity is shifted upward (since the main eddy moves upward) with increasing N, but its magnitude is only slightly decreased. Figure 4.15b shows that the $v$ velocity on horizontal centerline is suppressed with increasing N, and the boundary layer grows accordingly.

<table>
<thead>
<tr>
<th>y</th>
<th>N=0.25</th>
<th>N=0.50</th>
<th>N=1.00</th>
<th>N=2.00</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.10</td>
<td>$-5.54 \times 10^{-3}$</td>
<td>$8.78 \times 10^{-3}$</td>
<td>$2.34 \times 10^{-3}$</td>
<td>$1.38 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.20</td>
<td>$-1.18 \times 10^{-1}$</td>
<td>$8.84 \times 10^{-3}$</td>
<td>$9.28 \times 10^{-4}$</td>
<td>$1.57 \times 10^{-4}$</td>
</tr>
<tr>
<td>0.30</td>
<td>$-1.47 \times 10^{-1}$</td>
<td>$-1.07 \times 10^{-3}$</td>
<td>$1.69 \times 10^{-3}$</td>
<td>$-4.96 \times 10^{-4}$</td>
</tr>
<tr>
<td>0.40</td>
<td>$-3.62 \times 10^{-2}$</td>
<td>$-4.88 \times 10^{-2}$</td>
<td>$5.47 \times 10^{-3}$</td>
<td>$1.97 \times 10^{-4}$</td>
</tr>
<tr>
<td>0.50</td>
<td>$1.37 \times 10^{-2}$</td>
<td>$-1.60 \times 10^{-1}$</td>
<td>$2.46 \times 10^{-3}$</td>
<td>$4.44 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.60</td>
<td>$2.65 \times 10^{-2}$</td>
<td>$-1.08 \times 10^{-1}$</td>
<td>$-3.70 \times 10^{-2}$</td>
<td>$8.66 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.70</td>
<td>$2.73 \times 10^{-2}$</td>
<td>$-1.09 \times 10^{-2}$</td>
<td>$-1.43 \times 10^{-1}$</td>
<td>$-1.65 \times 10^{-2}$</td>
</tr>
<tr>
<td>0.80</td>
<td>$2.05 \times 10^{-2}$</td>
<td>$1.87 \times 10^{-2}$</td>
<td>$-8.44 \times 10^{-2}$</td>
<td>$-1.20 \times 10^{-1}$</td>
</tr>
<tr>
<td>0.90</td>
<td>$1.11 \times 10^{-2}$</td>
<td>$2.81 \times 10^{-2}$</td>
<td>$8.30 \times 10^{-3}$</td>
<td>$-8.60 \times 10^{-2}$</td>
</tr>
</tbody>
</table>

The velocity vectors in the central x plane ($x=0.5$) are plotted in Fig. 4.16a and Fig. 4.16b for the case N = 0 and N = 0.5, respectively. Figure 4.16a shows without the magnetic field, four vortex pairs are found at the bottom of the cavity and two vortices appears on the two top corners. These Taylor-Görtler-like vortex pairs have also been found by other researchers [22]. Figure 4.16b shows that when the magnetic field is increased to N = 0.5 all these bottom wall vortices disappear. This is obviously a result of suppressing the lower circulation in the bottom portion of the cavity.
Table 4.7: $Re = 3200 \, v$ velocity on horizontal line $y = 0.5$

<table>
<thead>
<tr>
<th>$x$</th>
<th>$N=0.25$</th>
<th>$N=0.50$</th>
<th>$N=1.00$</th>
<th>$N=2.00$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.10</td>
<td>$7.97 \times 10^{-2}$</td>
<td>$-2.62 \times 10^{-2}$</td>
<td>$-1.02 \times 10^{-2}$</td>
<td>$-2.57 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.20</td>
<td>$7.75 \times 10^{-2}$</td>
<td>$-7.82 \times 10^{-4}$</td>
<td>$-8.87 \times 10^{-3}$</td>
<td>$-2.42 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.30</td>
<td>$5.85 \times 10^{-2}$</td>
<td>$2.05 \times 10^{-2}$</td>
<td>$-6.79 \times 10^{-3}$</td>
<td>$-2.16 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.40</td>
<td>$3.50 \times 10^{-2}$</td>
<td>$3.98 \times 10^{-2}$</td>
<td>$-3.87 \times 10^{-3}$</td>
<td>$-1.69 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.50</td>
<td>$1.58 \times 10^{-2}$</td>
<td>$4.99 \times 10^{-2}$</td>
<td>$-3.07 \times 10^{-3}$</td>
<td>$-9.25 \times 10^{-4}$</td>
</tr>
<tr>
<td>0.60</td>
<td>$4.74 \times 10^{-3}$</td>
<td>$3.78 \times 10^{-2}$</td>
<td>$3.23 \times 10^{-3}$</td>
<td>$1.20 \times 10^{-4}$</td>
</tr>
<tr>
<td>0.70</td>
<td>$-3.10 \times 10^{-3}$</td>
<td>$-8.56 \times 10^{-3}$</td>
<td>$6.15 \times 10^{-3}$</td>
<td>$1.59 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.80</td>
<td>$-2.58 \times 10^{-2}$</td>
<td>$-8.28 \times 10^{-2}$</td>
<td>$7.10 \times 10^{-3}$</td>
<td>$3.14 \times 10^{-3}$</td>
</tr>
<tr>
<td>0.90</td>
<td>$-2.50 \times 10^{-1}$</td>
<td>$-4.85 \times 10^{-2}$</td>
<td>$7.25 \times 10^{-3}$</td>
<td>$4.15 \times 10^{-3}$</td>
</tr>
</tbody>
</table>
Figure 4.14: Streamlines in $z = 0.5$ plane, $Re = 3200$ for different $N$ (from left to right, top to bottom): $N = 0, 0.04, 0.0625, 0.09, 0.25, 0.5, 1.0, 2.0$
(a) $u$ velocity along vertical centerline  

(b) $v$ velocity along horizontal centerline

Figure 4.15: Centerline velocity in $z=0.5$ plane, $Re=3200$

(a) Velocity vector in middle $x$ plane, $N=0.00$  
(b) Velocity vector in middle $x$ plane, $N=0.50$

Figure 4.16: Velocity vector in middle $x$ plane for $Re=3200$
4.4.4 Simulations at Re = 5000

We finally consider an even higher Re of 5000. For this high Reynolds number we have used a 192³ grid and integrated the equations in time with high accuracy at each time step. For $N = 0$, the flow is unsteady. Steady solutions are obtained for $N = 0.25$ and higher. Figure 4.17a and 4.17b show the streaklines for $N = 0.25$ and $N = 0.5$. Since the qualitative features seen here are similar to those for Re = 3200 and Re = 2000, results for higher $N$ were not computed.

![Streamlines in z = 0.5 plane, Re = 5000 for N = 0.25 and N = 0.5](image)

(a) $Re = 5000$ and $N = 0.25$
(b) $Re = 5000$ and $N = 0.5$

Figure 4.17: Streamlines in $z = 0.5$ plane, Re = 5000 for $N = 0.25$ and $N = 0.5$

The ($x, y$) coordinates of the eye of the primary eddy in the symmetry plane ($z = 0.5$) for all Reynolds numbers are tabulated in Tab. 4.8 with different Stuart numbers. As seen in earlier plots with increasing strength of the magnetic field, the center of the primary eddy moves toward the top lid and the downstream corner (top and right corner). One may also notice that for $N = 2$, with increasing Reynolds number the eye of the primary eddy is closer to the right top corner (top and downstream side) of the cavity. For fixed Re, with increasing $N$ the path of the eye of the main eddies are nearly on a straight line. The upward motion of the eye of the main eddy is a result of the damping effect of the applied magnetic field.

Table 4.8: ($x, y$) coordinates of the eye of the primary eddy in $z = 0.5$ plane

<table>
<thead>
<tr>
<th>N</th>
<th>Re=400</th>
<th></th>
<th>Re=2000</th>
<th></th>
<th>Re=3200</th>
<th></th>
<th>Re=5000</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$x$</td>
<td>$y$</td>
<td>$x$</td>
<td>$y$</td>
<td>$x$</td>
<td>$y$</td>
<td>$x$</td>
<td>$y$</td>
</tr>
<tr>
<td>0</td>
<td>0.622</td>
<td>0.577</td>
<td>0.603</td>
<td>0.484</td>
<td>0.633</td>
<td>0.523</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>0.25</td>
<td>0.688</td>
<td>0.719</td>
<td>0.641</td>
<td>0.499</td>
<td>0.622</td>
<td>0.425</td>
<td>0.613</td>
<td>0.373</td>
</tr>
<tr>
<td>0.5</td>
<td>0.719</td>
<td>0.786</td>
<td>0.713</td>
<td>0.696</td>
<td>0.715</td>
<td>0.662</td>
<td>0.712</td>
<td>0.627</td>
</tr>
<tr>
<td>1</td>
<td>0.744</td>
<td>0.838</td>
<td>0.812</td>
<td>0.848</td>
<td>0.855</td>
<td>0.871</td>
<td>not computed</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.753</td>
<td>0.872</td>
<td>0.84</td>
<td>0.896</td>
<td>0.862</td>
<td>0.904</td>
<td>not computed</td>
<td></td>
</tr>
</tbody>
</table>
4.5 Summary

In this paper we have studied the flow of an electrically conducting fluid in a 3D cavity subjected to a magnetic field. The time-dependent Navier-Stokes equations are integrated with a second-order accurate numerical scheme and a very fine grid. The code is parallelized on a GPU for computational efficiency. The general features observed for the case of no magnetic field are in agreement with previous computational and experimental studies. The 3D simulations showed more flow structures including the growth of boundary layers from the side walls. The flow field parallel and close to the side walls is different from the flow in the symmetry plane due to the effect of the side walls. However, the flow behavior in the plane $z = 0.3$ is very similar to the flow field in the symmetry plane ($z = 0.5$). A unique aspect of 3D simulations is the prediction of the formation of TGL vortices in the spanwise direction. The applied external magnetic field is seen to suppress and delay the formation of these TGL vortices. With the magnetic fields, the flow fields are considerably modified as the magnetic field induced force brakes the momentum from the top wall shear. The main recirculating eddy is seen to shrink in size with the formation of a low-velocity region in the lower region of the cavity. With increasing value of the interaction parameter ($N$), the flow in the lower region of the cavity is nearly suppressed. Further, with a magnetic field, a nominally unsteady flow is stabilized and becomes steady. This paper presents the flow patterns and velocity profiles for 23 runs made at different Reynolds numbers and interaction parameters. We also provide tables of selected profiles of velocities for readers to benchmark their numerical/experimental results for the same parameters.

4.6 References


[7] S. Sundaresan, S. Nagarajan, S. Deshpande, R. Narasimha, 2D lid-driven cavity flow at high Reynolds...


[64] ANSYS-Fluent, 12.0 Documentation, Ansys Inc.
Chapter 5

Effects of EMBr and Submergence Depth on Flow in Mold Region

This chapter investigates the effects of EMBr and SEN submergence depth on the flow inside the SEN and the mold region of the caster. A parametric study is carried out using high fidelity LES model. The mathematical model and results presented in this chapter has been submitted for journal publication.

5.1 Introduction

Continuous casting produces over 95 pct of steel in the world today [1], hence even small improvements to this important industrial process can have large economic impact. The characteristics of turbulent flow in the mold region of the caster influences the creation of surface defects, slag entrainment and other issues related to the quality of steel. The velocity across the top surface of the mold is an important parameter affecting defect formation. A very small velocity causes reduced heat transfer and leads to hook formation, meniscus freezing, and other surface defects. On the other hand, if the top surface velocity is too large, the resulting turbulence and shear layer instability may entrain slag and form inclusions in the final product. Therefore, it is very important to choose operating conditions which produce flow patterns within an optimal operational window to avoid these problems. These operating conditions include: the mold cross section, casting speed, submergence depth, argon gas injection, and electromagnetic forces. The use of electromagnetic force is an attractive method to dynamically control the mold flow due to its easy implementation by simply changing the electric current in the coils of the device.

The electromagnetic forces are commonly generated by applying a magnetic field near the mold region. The magnetic field can be a static field attained by passing a DC current through electromagnets, or by a moving field using an AC current. When a DC current is applied to a coil, a static magnetic field is generated which in turn induces a Lorentz force field that acts against the flow. Thus this concept is usually referred to as an electromagnetic braking (EMBr) system. Based on the DC electromagnets shape and location, there are usually three types of static magnetic field configurations: local [2–7], single ruler [7–12] and double ruler [12–17]. The difference between these configurations as well as the use of AC in the electromagnets
are discussed elsewhere [18]. In this work we focus on the double ruler configuration which is widely used in industry and commonly known as the Flow Control Mold (FC-Mold) (ABB Automation Technologies). [19] In the double ruler configuration, two rectangular magnetic fields across the entire mold width are generated, with one positioned near the meniscus and the other below the nozzle ports [13–18]. This configuration is able to slow down [13, 16] or to speed up [12, 16] surface velocities in the mold region, and has been reported to decrease high-frequency turbulent fluctuations [12] and to improve meniscus stability [16, 17].

Owing to the hostile environment of the hot molten steel and the large number of design and operating variables which affect the process, it is difficult to optimize flow in continuous casting through measurements in real casters. Thus, computational models, water models [15] and physical models [7, 8, 11–14, 20, 21] with conducting fluids (e.g. GaInSn [8, 11, 12, 20, 21], tin [14] and mercury [7, 13] are used to help understand the process. Due to the rapid development of computer hardware, computational models are widely used [2–17] as tools to understand the physical phenomena, improve, optimize and control the process. Most computational models use the Reynolds Averaged Navier Stokes (RANS) approach and solve equations for the mean flow behavior [2, 4–6, 8, 10, 13, 14, 17]. Only a limited number of researchers have so far studied the transient behavior of the turbulent flow using well resolved [9, 16] large eddy simulations (LES) with EMBr [7, 9, 11, 12, 15, 16] or without EMBr [22].

The effects of the double-ruler EMBr system on the mean flow field in real commercial casters have been studied by several researchers. Early in 1993, Idogawa et al. [13] applied the RANS method to model experiments with mercury and showed that the double ruler system can reduce the mean velocity near the meniscus and the narrow face (NF). A RANS simulation of molten tin experiments indicated that the double ruler system with a maximum magnetic field of around 0.3T can effectively reduce the flow velocity in the bulk but can also cause very low velocities (around 0.032 m/s) at meniscus region [14]. Using the k-RANS model, Cho et al. [17] showed that the EMBr can stabilize the top surface by reducing surface level fluctuations by 50 pct.

As the transient flow behavior in the caster mold is quite important [23], several researchers have recently conducted transient LES simulations. Qian and Wu [9] showed the formation of transient unsymmetrical vortices even if the nozzle was placed at the center of the mold. They also found that the unsymmetrical upward recirculating flow in the mold can cause unsymmetrical surface flow and generate biased vortical flows on the top surface. However, a single ruler magnet at the meniscus was seen to suppress this vortex flow. Singh et al. [16] studied flow in a steel caster with a double ruler EMBr (max magnetic field 0.28 T), and showed that the EMBr makes the top surface flow more stable resulting in only small level fluctuations. The surface velocity was reduced from 0.6 m/s to ∼0.1 m/s. Recently, a vertical electromagnetic brake
system has been proposed and studied. In this system, two long vertical electromagnets are placed close to the narrow face and a steady magnetic field is passed through the thickness of the mold. In addition to reducing the surface velocity, this configuration also reduces the impact of the steel jet on the narrow face.

Accurate flow computations with electromagnetics can be challenging. Including the solid shell in the calculation has the effect of stabilizing the flow in the mold. Chaudhary et al. performed LES of a GaInSn model with single ruler and double ruler EMBr arrangements. Their study assumed the shell to be an electrically insulated wall. This led to the observations of a transient low-frequency large-scale oscillation in the mold and an unstable flow. Singh et al. modified this model of Chaudhary et al. by adding an electrically conducting shell. Their LES results showed that the conducting wall can help stabilize the flow and the unsteady low frequency oscillation flow seen previously can be suppressed.

All of the above studies showed that the magnetic field affects the time-mean flow behavior in the caster as well as the transient flow structure in the mold. It reduces the mean surface velocity and fluctuations. However, electromagnetic forces can also generate undesired effects if not used correctly in conjunction with other casting parameters (such as submergence depth). Thomas et al. investigated the combined effects of the local EMBr system and different submergence depths on the time mean flow using the RANS approach. Their results showed that even with same EMBr configuration, different submergence depths may considerably change the effectiveness of the braking system. However, less attention was paid to the combined effects of submergence depth and EMBr for the widely used double ruler system. Most work was focused on the effect of EMBr on steel flow in mold region without including the full SEN.

Although a few researchers have included the SEN in the computational domain, the magnetic field above the steel-slag interface was ignored. However, simulations with RANS have shown that a strong magnetic field modifies the mean flow also inside the SEN and reduces its turbulent kinetic energy. Therefore, it is clear that the transient simulations must include the effects of the magnetic field on the nozzle flow.

In the present work, we have studied the combined effects of submergence depth and EMBr on the turbulent flow in both the SEN and in the mold region of a real continuous caster using eight high resolution large eddy simulations. Both steady and transient flow fields are presented and the effects of changes in nozzle flow due to the EMBr are studied. In the following sections, the governing equations, details of the numerical model are given in Section 5.2. Section 5.3 describes results predicted by LES for a low casting speed of 1.3 m/min. Section 5.4 discuss the effects of SEN submergence depth and EMBr on the flow with a higher casting speed of 1.8 m/min. The mean and transient flow inside the SEN, inside the port, in the
mold region and region close to the top surface are discussed. The effects of EMBr on turbulence and free surface profile are investigated. Finally, the results are summarized in Section 5.5.

5.2 Governing Equations and Computational Model

5.2.1 Governing Equations for the Fluid Flow

In this work the turbulent flow in the mold is simulated by the technique of large eddy simulations (LES). The three-dimensional time-dependent Navier Stokes equations given below were solved:

\[ \nabla \cdot (\rho u) = s \]  

\[ \frac{\partial u}{\partial t} + \nabla \cdot (u \otimes u) = - \frac{\nabla p}{\rho} + \nabla \cdot [(\nu + \nu_{sgs})(\nabla u + \nabla u^T)] + \frac{F_L}{\rho} + S_{sink} \]  

where \( \rho \) is the density of molten steel, \( s \) and \( S_{sink} \) are the mass and momentum sink terms[25–27] added to include the effect of the solidifying shell, \( u \) is the velocity vector, \( p \) is a modified static pressure which includes the normal stresses, \( \nu \) is the kinematic viscosity of the molten steel and \( \nu_{sgs} \) is the eddy viscosity that represents the subgrid stress. The outer product \( u \otimes u \) is equivalent to a matrix multiplication \( uu^T \).

In this study, \( \nu_{sgs} \) is modeled by the Coherent-structure Smagorinsky model (CSM) sub-grid scale (SGS) model[28]. This model has been successfully tested and used previously in predicting fluid flow in steel casters with magnetic fields[11, 16]. A brief introduction of the CSM model is provided in Section 5.2.2, and more details are available elsewhere [28]. The term \( s \) in the continuity equation is a sink term due to the solidifying shell, and the details about its implementation and the shell profile are given in Section 5.2.3. The imposed magnetic field affects the fluid flow through a Lorentz force field source term \( F_L \) in the momentum equation. The governing equations for computing \( F_L \) are discussed in Section 5.2.4. Section 5.2.5 discusses the numerical method and the performance of the multi-GPU code. Section 5.2.6 presents the geometry of the caster, the computational domain, mesh, boundary conditions and the computational details.

5.2.2 The CSM Sub-grid Scale Model

In LES, the eddy viscosity \( \nu_{sgs} \) is used to model the influence of the turbulent scales that are not resolved explicitly. In this work the \( \nu_{sgs} \) is calculated using a coherent structure function given by the CSM sub-grid scale model[28]. The CSM model does not need a wall-damping function. In CSM, first the Smagorinsky
eddy-viscosity model is used to compute $\nu_{sgs}$ as

$$\nu_{sgs} = (C_s \Delta)^2 \sqrt{2||S||}$$  \hspace{1cm} (5.3)$$

where $C_s$ is the Smagorinsky constant and $\Delta$ is the cell size and $S$ is the rate-of-strain tensor given by $S = 1/2 (\nabla u + \nabla u^T)$. In the CSM model, $C_s^2$ is calculated locally by the following:

$$C_s^2 = C_{csm} |Q/E|^{3/2} \left(1 - Q/E\right)$$  \hspace{1cm} (5.4)$$

$$Q = 1/2 \left(||W||^2 - ||S||^2\right)$$  \hspace{1cm} (5.5)$$

$$E = 1/2 \left(||W||^2 + ||S||^2\right)$$  \hspace{1cm} (5.6)$$

Where $C_{csm}$ is a model constant equal to 1/22 and $W = 1/2 (\nabla u - \nabla u^T)$ is the vorticity tensor (also known as rate of rotation tensor). The CSM model appropriately damps the eddy viscosity in wall boundary layer regions and also automatically incorporates the effect of anisotropy induced by the applied magnetic fields on the subgrid-scales[29]. Therefore, no additional modifications to account for anisotropic subgrid effects are needed or added.

### 5.2.3 Modeling of the Solidifying Shell

During the casting process, liquid steel continuously solidifies as it crosses the liquid / solid interface which defines the shell. The shell thickness grows as the metal moves downwards in the caster causing the cross-section of the liquid metal region to shrink, and the liquid steel mass flow rate to decrease as the phase change progresses from liquid to solid. This reduction in volume of the molten steel is included in our model by imposing the shell profile as the domain boundary and applying a mass sink term in the finite volume cells adjacent to the shell boundary.

The growth of the shell thickness depends on the rate of heat extraction into the mold walls. In this work a pre-calculated steady shell profile is applied. The assumed shell thickness $s$ (mm) at any point $z$ m below the meniscus is plotted in Figure 5.1 and it is calculated as:

$$s = k\sqrt{z/V_c}$$  \hspace{1cm} (5.7)$$
where \( V_c \) denotes the casting speed (m/s) and the constant \( k = 3 \text{ mm/s}^{1/2} \) is chosen to match a break-out shell profile from the caster at Baosteel.

A stair-step representation of the shell is used for computational efficiency, considering that the mesh is very refined. Mass and momentum sink terms are added to the cells near the shell to include the effects of the loss in mass due to the solidification. These sink terms are added only in those molten steel cells whose bottom neighbor cell is in the domain of solid shell region as shown in Figure 5.2. The amount of the mass that leaves the specific cells per second due to solidification is \( \rho V_c \Delta x \Delta y \), where \( \Delta x \) and \( \Delta y \) are the cell dimensions in the \( x \) (mold width direction) and \( y \) (mold thickness direction) directions, respectively. The momentum associated with this lost mass is subtracted as a momentum sink \( \dot{s} = \rho V_c \Delta x \Delta y V_c^{-1} \).[25]

Figure 5.1: Shell thickness profile at two casting speeds used in the computational model

Figure 5.2: Schematic of grid near solid shell showing a cell with non-zero mass sink terms
5.2.4 Equations for MHD Fields and Forces

In this work, an electric potential method is used to compute the Lorentz force. This method exploits the fact that the induced magnetic field is much smaller than the externally-imposed magnetic field and therefore can be ignored. [4, 5, 8–14, 16, 24] When a magnetic field $B$ is applied onto a moving conducting material (with electrical conductivity $\sigma$), an electric field is created and the current density $J$ can be computed through the Ohm’s law given by equation (5.8). The current conservation law is given by equation (5.9). The electric potential $\Phi$ satisfies a Poisson type equation (5.10) from which $\Phi$ can be solved. The Lorentz force is given by equation (5.11).

$$J = \sigma \left( -\nabla \Phi + \mathbf{u} \times \mathbf{B} \right)$$  \hspace{1cm} (5.8)

$$\nabla \cdot J = 0$$  \hspace{1cm} (5.9)

$$\nabla \cdot (\sigma \nabla \Phi) = \nabla \cdot \left[ \sigma (\mathbf{u} \times \mathbf{B}) \right]$$  \hspace{1cm} (5.10)

$$\mathbf{F}_L = J \times \mathbf{B}$$  \hspace{1cm} (5.11)

Note that both the molten steel and the solidified shell are conducting materials, so equations (5.8) to (5.11) must be solved in both the liquid steel and shell regions. Replacing the shell with an electrically-insulated wall boundary condition leads to incorrect results.[11, 12] Thus, in this work, the above MHD equations are solved in the entire domain including the solid shell region. An insulated wall is applied at the exterior of the shell, considering that the slag in the mold / shell gap has very low conductivity.

5.2.5 Numerical Method and Code Validation

Previous LES simulations of steel caster flow fields have been limited by computational resources, so only relatively coarse grids were used and parametric studies were not feasible. In the present work, to accelerate computation speed, the governing equations are solved on multiple Graphics Processing Units (GPU). A in-house code CUFLOW[30–34], written in CUDA Fortran language, has been extended to multiple GPUs in parallel through the Massage Passing Interface (MPI)[33]. This multi-GPU code has high parallel efficiency and exploits larger GPU memory. CUFLOW has been validated in studies of Magnetohydrodynamic flow in a square lid-driven cavity[35], generalized Newtonian fluid flow[36], bubble rise in duct[37, 38], and flow in the mold region of continuous casters of steel with and without EMBr.[11, 12, 16]
The calculations in this paper were performed on the GPUs of Blue Waters supercomputer at the National Center for Supercomputing Applications, Illinois. The computational domain was decomposed in the casting direction and uniformly distributed onto six Nvidia Kepler K20x GPUs. A fractional step method is used to solve the continuity and momentum equations. Complete details of the solution algorithm are given elsewhere.[31–33] The most computationally intensive parts are the solution of the two Poisson equations (pressure-Poisson equation and electrical-Poisson equation). In this work, the two Poisson equations are solved efficiently on GPUs by a V-cycle multigrid method, and Red-Black Successive Over Relaxation (SOR) with over-relaxation parameter of 1.6.

5.2.6 Computational Domain, Mesh, Boundary Conditions and Computational Cost

This work studies flow in the No. 4 steel caster in Baosteel, Shanghai, P. R. China. The casting parameters, flow domain and steel properties are given Table 5.1.

Table 5.1: Process parameters (*: Casting conditions when plant measurements were carried out)

<table>
<thead>
<tr>
<th>Process parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mold thickness ($L_t$)</td>
<td>230 mm</td>
</tr>
<tr>
<td>Mold width ($L_w$)</td>
<td>1200*: 1300 mm</td>
</tr>
<tr>
<td>Slide gate opening area fraction</td>
<td>70 pct*: 80 pct</td>
</tr>
<tr>
<td>SEN Submergence depth</td>
<td>170 mm or 180 mm</td>
</tr>
<tr>
<td>Nozzle port downward angle</td>
<td>15°</td>
</tr>
<tr>
<td>Nozzle port area (width $\times$ height)</td>
<td>65$\times$83 mm²</td>
</tr>
<tr>
<td>Casting speed ($V_c$)</td>
<td>1.3 or 1.8 m/min</td>
</tr>
<tr>
<td>Steel density ($\rho$)</td>
<td>7000 kg/m³</td>
</tr>
<tr>
<td>Argon density ($\rho_p$)</td>
<td>0.5 kg/m³</td>
</tr>
<tr>
<td>Steel dynamic viscosity ($\mu$)</td>
<td>0.0063 kg/(m·s)</td>
</tr>
<tr>
<td>Liquid Steel electrical conductivity ($\sigma$)</td>
<td>714000 S/m</td>
</tr>
<tr>
<td>Solid Steel electrical conductivity ($\sigma$)</td>
<td>787000 S/m</td>
</tr>
</tbody>
</table>

A 3D view of the liquid steel region of the computational domain is shown in Figure 5.3. A Cartesian grid with approximately 16 million finite volume cells (cell size 4 mm) was used to discretize the computational domain which included the slide gate, SEN and mold region of the caster. The domain extends to 2.6 m below the meniscus. To incorporate the effect of the solidified shell on the electrical current and Lorentz force distributions, the solid shell is included in the computational domain by adding solid cells on the sides of the liquid region. In these cells, the fluid flow equations are not solved, but only the MHD equations are solved. For these cells, the velocity component in the casting direction ($z$ direction) is prescribed to be the casting speed, with other velocity components being zero. The rate of flow of the molten steel through the SEN into the mold is controlled by the slide-gate that moves between the geometric center and the Inside
Radius (IR) of the caster, as indicated in Figure 5.4. To get a casting speed of 1.8 m/min, the slide gate was kept 80 pct open and the molten steel flow first entered the inner radius region of the SEN.

The fluid velocity at the SEN inlet is fixed at 1.66 m/s to match total the mass flow rate of the steel. At the outlet, zero derivative boundary conditions are applied. All other domain boundaries and the interface between the liquid steel and solidified shell are prescribed as no slip and no penetration conditions. As mentioned earlier, the MHD equations, an insulated boundary condition is applied at the outer most boundary of the computational domain and the shell is considered electrically conducting.

This works investigates turbulent flow with the double ruler EMBr system. The induced magnetic fields
for three different configurations of the coil current were measured at Baosteel\[39\]. Figure 5.5 shows the magnetic field strengths measured at discrete points and the interpolated magnetic field that is used in the calculations. The top surface (molten-steel / slag layer interface) is at \( z = 0 \), and the location of the submerged nozzle ports is shown as a shaded region. All three current levels of the upper ruler have very weak magnetic field strength across the ports, similar increasing strengths below the ports, and increasing strengths above the ports, reaching different peaks, depending on the coil current. With the coil current of 400A and 850A, the maximum magnetic field strength near the meniscus is 1.8\( \times \) and 2.5\( \times \) larger than that with zero top coil current. Note that with a double ruler system, even when the top coil has zero current, there is still a local peak magnetic field (-0.14 T) generated near the top electromagnet. Thus, the field profile is different from that of a single ruler system, positioned at the lower electromagnet location. The measurements show that the variations of the magnetic field along the mold thickness and width are very small in all three cases, so the imposed magnetic field only varies in the casting direction (\( z \) direction).

As listed in Table 5.2, eight LES simulations were carried out on the Blue Waters supercomputer. All simulations were started with zero initial velocity, and the flow fields were allowed to develop for 15 seconds of real flow time. After that, turbulence statistics were collected for \( \sim 25 \)s until the simulation finished at \( \sim 40 \) seconds. Specifically, velocity and pressure were stored in selected slices (Figure 5.6). The selected slices included the symmetry center planes \( (x = 0; \ y = 0) \), the port exit planes \( (x = \pm 0.047m) \), and a parallel plane 1 cm below the top surface \( (z = 0.01m) \). A time step around 0.0002 second was used for stability. At each time step, convergence was defined when the pressure residual reduced by four orders of magnitude, reaching \( \sim 1 \times 10^{-8} \). Typically on six GPUs about two days (144 node hours) were required to complete a

Figure 5.5: Measured magnetic field showing measurement points (dots) and interpolated (lines) used in model
40 seconds LES simulation on a grid with ~ 16 million cells.

Figure 5.6: Location of important surfaces

5.3 Model Validation

The surface velocity of the molten steel in the mold is critical to the final quality of the product. Various techniques such as nail-board dipping method[40] and Sub-meniscus Velocity Control (SVC) devices[41, 42] have been previously used to measure the surface velocity. In this work, a rod-deflection device (similar to a previous inclination method[43]) was used to measure the surface velocity. Figure 5.7 shows a schematic of this device. The test consists of measuring the deflection angle of a rod (probe) about its pivot point while its other end is dipped into the molten steel. The molten steel flow impinges on the rod and generates a drag force which balances the weight of the rod after it rotates an angle of . This angle is then converted into a velocity parallel to the Wide Face (WF) of the mold. These measurements were performed at Baosteel No. 4 caster with a strand size of 230 mm × 1200 mm, casting speed of 1.3 m/min and a 70 pct slide gate opening. No argon gas was injected and the EMBr system was turned off during the measurements. The device probe was placed between the SEN and NF (as shown in Figure 5.8) and the measured steel surface velocity was recorded every 5 seconds during a seventy second interval. To compare with the plant measurements, a LES
Figure 5.7: Velocity measurement device

Figure 5.8: Comparison of predicted surface velocity (centerplane, quarter mold, 1 cm below top surface) with plant measurements
simulation with a grid of 15.5 million cells was conducted for 80 seconds with the same casting conditions as the measurements. The $x$ direction velocity (parallel to WF) calculated at the probe location of 1 cm below the top surface and at the quarter mold region in the middle plane is compared with the measurement in Figure 5.8. Both the LES simulations and the measurements show that the flow direction is from NF to the SEN, with an average velocity around 0.2 m/s. Also, both show that the surface velocity sometimes drops to a relatively low value of 0.12 m/s for 10-20 seconds, and at other times it can be twice as big. The velocity changes with a low frequency of 0.025 to 0.05 Hz. This agreement between the computations and measurements gives confidence in the LES approach.

5.4 Simulations at Low Casting Speed

5.4.1 Flow in the SEN and SEN Port

The flow in the SEN and the port are next investigated. Figure 5.9(a) shows contours of the time-averaged vertical velocity $\bar{u}_z$ in the symmetry plane of the SEN. The high velocity jet flowing the slide-gate opening first enters the inner radius side of the SEN with a maximum velocity slightly above 2 m/s. The jet is then deflected off of the SEN side wall and flows stronger down the outer-radius wall side, forming a strong counterclockwise swirling flow at the bottom of the SEN. Close to the slide gate, a 0.3 m long recirculation

![Figure 5.9](image)

Figure 5.9: (a) Contours of time-averaged $\bar{u}_z$ velocity in centerplane of SEN; (b) snapshot of transient $u_z$ at $t = 64.8s$
zone starting inside the sliding plate forms and occupies almost half of the SEN diameter. This large recirculation region and the bottom swirl are also seen in Figure 5.9(b) which shows an instantaneous vertical velocity field at \( t = 64.8 \) second. However, this snapshot shows that the jet bounces several times before it reaches the bottom of the SEN.

Figure 5.10(a) shows contours of the time-averaged \( x \) velocity (\( \bar{u}_x \)) at the port exit plane (see location in Figure 5.6). The large counterclockwise swirl structure observed in the SEN bottom (Figure 5.9) extends through the port. In Figure 5.10, negative \( \bar{u}_x \) means flow is leaving the SEN and entering the mold, while a positive \( \bar{u}_x \) indicates flow reversal going back into the port from the mold as a back-flow region. On average, the flow leaves the SEN closer to the outer radius side of the port, with an average velocity of 1 m/s. Two back-flow regions are found inside the port: one near the middle (biased towards the inner radius side) and the other at the top and outer radius side of port. The back-flow can make the jet unstable and substantially alter the mold flow pattern.\[44, 45\] This back-flow regions together occupy about one quarter of the entire port area. The middle back-flow region formed due to the strong swirl structure, which generates a local low pressure region that sucks fluid into the center of the swirl. A snapshot of flow in the port shown in Figure 5.10(b) indicates that the maximum velocity leaving into the mold is around 1.5 m/s and the back-flow is as large as 1 m/s.

A snapshot of the iso-surface of constant \( u_x \) velocity is shown in Figure 5.11 and illustrates the complex 3D structure of the jet that enters the mold region. The jet enters the mold with a higher velocity near the top outer-radius corner of port. At that corner, and in the middle of the port, the back-flow regions are seen (with yellow color). The structure of the jet leaving the port is caused by the swirl residing at the bottom.
5.4.2 Flow in the Mold Region

The flow in the middle plane of the mold and top surface is shown in Figure 5.12. Figure 5.12(a) shows at one centimeter below top surface that the flow direction is mainly from NF to SEN in most regions, and the maximum mean velocity magnitude is about 0.2 m/s near the quarter mold width. Close to the SEN, the mean flow speed is smaller than 0.1 m/s. It is necessary to note that the instantaneous flow speed close to the SEN often changes directions and is usually much higher than the mean value. The low average speed shown is mainly caused by cancellation of the different instantaneous flow directions. At regions very
close to the NF, the flow direction is towards NF, which is caused by the small secondary circulation region at the corner of the mold. However, the flow is very weak with a mean flow speed of less than 0.1 m/s. Figure 5.12(b) shows the flow streamlines in the middle plane of the mold. As expected, the time-averaged flow is symmetric about the middle $x$ plane ($x = 0$) and the jet has higher speed at top and bottom of the ports. The typical double-roll flow pattern is seen in the mold region.

The swirl direction inside the port is also important and several previous works have discussed the relation of the swirl flow in the SEN with the bulk flow in the mold.[44, 46] In this case, the swirl at the bottom of the SEN has three different patterns as shown in Figure 5.13: (1) two symmetrical swirls of almost equal size at the bottom of the SEN; (2) one generally-clockwise swirl that rotates from inner radius side to outer radius side across the nozzle bottom; (3) one counter-clockwise swirl that rotates from outer radius side to inner-radius side. Figure 5.14 shows the swirl types at different times of the simulation and the corresponding top surface velocity in the mold. For this low casting speed, the predominant swirl direction is counterclockwise, generated from stronger flow down the outer radius, crossing the bottom to the inner radius. This is counter intuitive because recall the steel flow first enters down the inner radius side of SEN. However, the jet enters the SEN from the slide gate at an angle, which causes the core of the jet to bounce off the inner radius towards the outer radius of the SEN inner wall. Thus the swirl direction is not only related to the position of the slide gate (which varies with casting speed), but is also depends on the SEN length. The time histories of $u_z$ velocity component at the center plane, quarter mold region and 1 cm below top surface are shown in Figure 5.14. In these histories, no significant relation is seen between the top surface velocity and the swirl direction. This is likely due to the short periods ($< 2s$) of clockwise swirl, which are much shorter than the recirculation time in the upper roll.

Figure 5.13: Swirl at bottom of SEN (low casting speed of 1.3 m/min)
5.5 Simulation at High Casting Speed

5.5.1 Effect of Submergence Depth on Flow in SEN Ports

Several simulations were next conducted at high casting speed of 1.8 m/min to study the effects of submergence depth on the flow in the caster. Submergence depth changes regularly during the plant operation, in order to lessen the effects of refractory erosion at the interface with the liquid slag and steel top surface. Modifying submergence depth may change the flow pattern in the mold and affect the top surface velocity, level fluctuations and meniscus heat transfer.[47] When an EMBr system is used, the submergence depth affects the mold flow as well as the EMBr efficiency dramatically. [18] Figure 5.15 shows the direction of the swirl rotation at the bottom of SEN and the $x$ direction velocity at top surface region. With the higher...
casting speed, two symmetrical counter-rotating swirls persist for most of the time. The top surface velocity at quarter mold region is increased from \( \sim 0.2 \text{ m/s} \) (at \( V_c = 1.3 \text{ m/min} \)) to \( \sim 0.385 \text{ m/s} \) (170 mm submergence) or 0.325 m/s (200 mm submergence). Increasing the submergence depth by 17 pct is seen to cause a 15 pct decrease in top surface mean flow velocity. However, the surface level fluctuations are less affected by this change in submergence depth. A low frequency (\( \sim 0.03-0.04 \text{ Hz} \)) variation of the top surface \( x \) direction velocity is seen for different submergence depths. However, this low frequency variation is not seen for the other two velocity components (\( u_y \) and \( u_z \)).

Figure 5.16 shows contours of time-averaged \( x \) velocity (\( \bar{u}_x \)) (in/out of port) in a cross section through the port at \( x = -0.45 \text{ m} \) for different submergence depths. The black line is where \( \bar{u}_x = 0 \). Inside and at the upper portion of the port, a back-flow region is seen. This region occupies about 15 pct of the total port cross section area, with an inward velocity \( \bar{u}_x \) less than 0.5 m/s. This is caused by the jet entering the mold region from the lower portion of the SEN port. Two small swirls are seen exiting the bottom of the port. The swirling steel flow also forms a low pressure region in the center low-velocity region of each swirl, which sucks in fluid from the surroundings.

### 5.5.2 Effect of Submergence Depth on Mold Flow

Contours of time-averaged velocity magnitude \(|\bar{u}|\) in the middle plane of the mold and 1 cm below top surface are shown in Figure 5.17 for two different submergence depths with no EMBr. The stream traces in the middle center plane show a typical double role flow pattern for both submergence depths. Because of the high resolution in the LES approach, smaller recirculation zones at the corner between narrow face and top surface were resolved. Each small recirculation zone spans \( \sim 70 \text{ mm} \) and rotates with a smaller downward
velocity (0.05∼0.1 m/s) at the side close to the narrow face. This downward velocity may increase the chance of slag entrainment and capture of the inclusion particles as well as argon bubbles. [48, 49]

Near the top surface, the fluid moves almost parallel to the wide face with a speed of ∼0.35 m/s without any cross flow. This is slightly higher than suggested optimal surface velocity ranges: higher than 0.1∼0.2 m/s[50] and less than 0.3 m/s [51] ∼0.4 m/s[50]. A lower surface velocity causes inadequate heat transfer to melt the slag powder and leads to entrapment of slag particles and freezing of the meniscus. A higher velocity on the other hand may cause shear instabilities and level fluctuations which can entrain slag and causes other surface defects.[39, 49, 51, 52] The contour lines with a velocity magnitude of $|\vec{u}| = 0.4$ m/s and $|\vec{u}| = 0.1$ m/s are marked with thicker black and red lines, respectively. Higher velocity regions are found close to the outer radius side of the wide face. This biased flow velocity shows that although the slide gate asymmetry may not lead to a cross flow near the meniscus region, it may cause stronger flow at the outer radius side. However, the time-averaged values can be misleading because they do not show regions of large flow fluctuations, where changing flow directions cancel their effect during computing the mean velocity. A snap-shot of the flow close to the SEN is shown in Figure 5.18, which shows locations near the SEN with local velocity magnitude higher than 0.4 m/s. These high-velocity vortices can result in slag entrainment into the molten steel and defects in the final product if captured by the shell.[40]

5.5.3 Effect of EMBr on Flow in SEN

Previously[16, 17], the double ruler EMBr has been shown to be able to stabilize the flow in the mold region and to reduce surface fluctuations. However, previous studies have not noted the effect of EMBr on turbulent
flow inside the nozzle, and its effect on the swirling flow exiting the ports. Many previous numerical models did not include the geometry of the full SEN [7–15, 22, 24] or simply applied a zero magnetic field inside the SEN.[16] However, plant measurements indicate that the magnetic field generated by the top ruler is strong inside the SEN, and only decays to zero close to the slide gate. Therefore, a strong transverse magnetic field typically exists through a large portion of the nozzle, and influence nozzle flow and jet behavior. Figure 5.19 shows the time-averaged $z$ velocity $\bar{u}_z$ in the symmetry plane ($x=0$) of the SEN with casting speed of 1.8 m/s. It is seen that the flow separates when it exits the middle sliding gate and reattaches about 0.2 m below the separation point. A recirculation region of 0.2 m long and almost half the SEN diameter forms. This

Figure 5.18: Snapshot of instantaneous velocity at top surface (no EMBr)

Figure 5.19: Time-averaged flow in SEN symmetry plane ($x = 0$) under different magnetic field. Contour of $\bar{u}_z$
large recirculation region can cause clogging as well as aspirating in gas due to its low pressure. Because of
the weak magnetic field near the slide-gate region, reattachment not significantly altered, but the velocity
inside the recirculation zone is slightly reduced. As magnetic field strength increases down the SEN, the
flow is reattached and the jet slows down. The jet penetration length, defined as the distance from the slide
gate to where the maximum jet velocity drops to 2.6 m/s, becomes smaller. Two counter-rotating swirls are
usually seen at the bottom of the SEN. When the top coil current increases to 850 A, however, only one tiny
swirl region is observed at the inner radius side of the nozzle.

Figure 5.20 shows the time averaged $\bar{u}_z$ velocity in the middle plane ($y = 0$) of the SEN for different
magnetic field strengths. It can be seen that close to the slide gate, the maximum velocity can be as large
as 4 m/s. With a strong magnetic field, higher velocity layers are seen close to the SEN wall in the lower
part of the SEN. The jets leaving the SEN ports also become thinner with increasing strength of the applied
magnetic field.

To reveal the thin high-velocity regions in the lower SEN caused by EMBr, Figure 5.21 plots the time-
averaged $\bar{u}_z$ velocity across the SEN at 0.15 m below the liquid level in the mold. An “M-shaped” velocity
profile is seen with EMBr, with the velocity peak increasing with field strength. The maximum streamwise
velocity in the M-shaped profile is around 2.8 m/s for the strongest magnetic field of the top coil, with
current of 850 A. This M-shaped profile is commonly seen in channel and duct flows$^{[53, 54]}$ when a strong
transverse magnetic field is applied. However, to our knowledge, this is the first study to report an M-shaped
profile in the SEN of a continuous caster. As the magnetic field strength increases, the M-shaped profile becomes steeper and forms at higher locations inside the SEN. This M-shaped profile has higher momentum near the SEN side walls. As the jets exit the SEN, the high velocity layer is bent by the bottom angle of the port. However, its high momentum causes the jet to penetrate the model region as a thinner and stronger jet leaving the bottom of the port. To satisfy local mass conservation, a larger back-flow region inside the upper port is expected. These findings show that it is important to including the magnetic field inside the SEN when studying flow in the mold with EMBr.

5.5.4 Effect of EMBr on Swirl Flow in the Ports

A strong swirling flow extends across the SEN bottom and through both ports, which is caused by the asymmetric flow down the nozzle. In Figure 5.22, time-averaged y direction velocity ($\bar{u}_y$) contours are shown, which reveal this swirling region. Increasing the magnetic field strength reduces the size and strength of this swirl. With a magnetic field, the swirl is confined to only a small region near the central axis, and flow exiting the ports has no swirl.

Figure 5.23 shows the effect of magnetic field strength on the time-average flow at port exit. The sizes of the back-flow region, where flow enters the upper port, are similar to those at $V_c = 1.3$ m/min. Figures 5.23(a) and (b) indicate that with zero top coil current, the back-flow region in the port is much smaller. With a top coil current of 850 A, the back-flow region is twice as large. The thinning effect of the EMBr on the jet leads to a larger back-flow region in the port, exceeding one third of the port area. As the total volume flow rate is constant, the average speed of the steel jet entering the mold must increase with EMBr.

The EMBr affects the flow in SEN, which in turn affects swirl in the SEN bottom and ports. The number of swirl regions and their rotation direction(s) are shown in Figure 5.24 for different magnetic field
Figure 5.22: Velocity at bottom of the SEN. Positive $y$ velocity means flow exiting port (a) No EMBr; (b) bottom coil current 850 A; (c) top coil current 400 A and bottom coil current 850 A; (d) both coils have current of 850 A

Figure 5.23: Contours of time-averaged $x$-velocity in port (positive $x$ velocity means back flow) for $d_{sub}$ = 170 mm with different EMBr: (a) without EMBr (b) bottom coil current 850A; (c) top coil 400 A and bottom coil 850A; (d) both coils 850 A
strengths and submergence depth of 170 mm. In contrast with the low casting speed cases, where a single swirl rotating from outer radius to inner radius was seen, these cases are more symmetrical with two swirl regions. This is likely due to the larger slide gate opening under higher casting speed. Increasing the top coil current to 850 A, the maximum magnetic field at the meniscus region reaches \( \sim 0.34 \) T, and the swirl almost disappears, as shown in the “tiny swirl” inset. Also, for these cases, no significant relation between the top surface velocity and the swirl is seen. Turning on the magnetic field, the top surface velocity is seen to reduce from \( \sim 0.4 \) m/s to less than \( \sim 0.06 \) m/s, with the surface becoming very stable.

5.5.5 Mold Flow

Figure 5.25 shows the stream traces superposed with the contours of time-averaged velocity magnitude in the middle center plane of the mold. The well-known double roll flow pattern is seen for all cases. With EMBr, the circulation regions are closer to the jets, in agreement with previous studies.[16] Between the narrow face and the main recirculation zone that above the jet, a small counter-rotating circulation region is seen on each side. As EMBr field strength increases, the jets exiting the SEN ports become thinner and stronger.

5.5.6 Surface Flow and Level Fluctuations

As shown earlier, the two jets entering the mold region form classic double-roll recirculation zones. The strength of these recirculation regions strongly affects the top free surface behavior. Figure 5.26 shows the time-averaged velocity magnitude 1 cm below the top surface for the three cases with EMBr, and the
Figure 5.25: Time averaged velocity magnitude $|\bar{u}|$ in middle plane of the mold

Figure 5.26: Contours of time-averaged velocity magnitude at 1 cm below top surface (a) bottom coil current 850 A; (b) top coil current 400 A and bottom coil current 850 A; (c) both coils have current of 850 A
centerline x-velocities ($\bar{u}_x$) are shown in Figure 5.27. For all three cases, surface velocity less than 0.07 m/s, and near the narrow face the velocity is less than 0.01 m/s. This low surface velocity is inadequate to deliver enough superheat to the meniscus region, thus leading to meniscus solidification, hook formation and entrapment of slag or inclusion particles.

![Figure 5.27: Mean x-velocity across center line 1cm below top surface. (Legend shows current (A) in top and bottom coils)](image)

With EMBr, the top surface velocity profile is much slower and more symmetric about the $y = 0$ plane. The strong surface velocity and the small secondary vortice near the NF, seen in the case without EMBr, both disappear with EMBr. With top coil current of 850 A, the high velocity region is slightly larger than with the weaker top magnetic field. The reason is that the jets are thinner and much stronger when the top magnetic field is strong.

Flow across the top surface also affects the surface level profile and its fluctuations. Large surface level fluctuations may cause defects in the product by entraining slag into the molten steel.[55] The surface-level can be approximated as follows[56]:

$$h = \frac{p - p_{mean}}{\rho g} \quad (5.12)$$

where $p_{mean}$ is the time averaged pressure along the center line and 1 cm below the top surface, as shown in Figure 5.28(a). Figure 5.28(b) shows the surface-level profiles calculated using equation (5.12). Without EMBr, the surface level has a “W” shape with highest surface level of +5mm near near the narrow face and the lowest level of -3.5 mm at the quarter-mold region near P8. When a magnetic field is applied, the surface level profile reverses to a “U” shape. This “U” shape profile has its highest surface level at the narrow face and lowest level near the SEN. With a stronger magnetic field, the difference, $\Delta h$, between the highest and lowest surface levels increases. Figure 5.29 shows similar trends with a submergence depth of 200 mm.
As expected, increasing the submergence depth reduces the maximum surface level difference. Specifically,

without EMBr, increasing submergence depth from 170 mm to 200 mm reduces $\Delta h$ from 8 mm to 5 mm.

Figure 5.30 shows the time history of the surface level at three typical points: P7, P8 and P9 (with locations given in Figure 5.28(a)). The standard deviation of the surface level, $\sigma_h$, indicates the time-average level fluctuations and are shown in Table 5.3. Without EMBr, the instantaneous surface level fluctuations can be as large as $\sim10$ mm (see Figure 5.30(a)). With just the bottom coil current of 850 A, the surface level fluctuations are only $\sim5$ mm near the SEN and 2-3 mm at the quarter mold and narrow face regions. Further applying the current in the top EMBr ruler coil, the surface level fluctuations drop to $\sim2$ mm. Without EMBr, the average fluctuations $\sigma_h$ are $\sim2$ mm. Increasing the magnetic field strength reduces the
Table 5.3: The standard deviations of surface level $\sigma(h)$ (mm)

<table>
<thead>
<tr>
<th>EMBr Coil Current (A)</th>
<th>Point P7</th>
<th>Point P8</th>
<th>Point P9</th>
</tr>
</thead>
<tbody>
<tr>
<td>No EMBr</td>
<td>1.9</td>
<td>1.5</td>
<td>2</td>
</tr>
<tr>
<td>Itop 0 Ibot 850</td>
<td>1</td>
<td>0.7</td>
<td>0.6</td>
</tr>
<tr>
<td>Itop 400 Ibot 850</td>
<td>0.6</td>
<td>0.5</td>
<td>0.5</td>
</tr>
<tr>
<td>Itop 850 Ibot 850</td>
<td>0.2</td>
<td>0.2</td>
<td>0.3</td>
</tr>
</tbody>
</table>

level fluctuations. These results quantify how EMBr can be used to lessen the surface level profile variations as well as to reduce the surface level fluctuations.

Figure 5.30: Surface level history at (a) point P7, (b) point P8, and (c) point P9 with different magnetic field strengths (see Fig. 5.28 for locations. Legend shows current in top and bottom coils in Amps).

5.5.7 Turbulent Kinetic Energy

To investigate the effect of EMBr on turbulent flow, the Turbulent Kinetic Energy (TKE) is computed. TKE represents the strength of the turbulence in the flow and can be computed by

$$TKE = \frac{1}{2} \left[ \langle u_x'^2 \rangle + \langle u_y'^2 \rangle + \langle u_z'^2 \rangle \right]$$

where $u_x'$, $u_y'$ and $u_z'$ are the fluctuating velocities in the $x$, $y$ and $z$ directions, respectively. The angled brackets imply ensemble averaging in time. Figure 5.31 shows the predicted TKE in the middle plane of the mold. Since the predicted TKE is roughly symmetric to the $x = 0$ plane, only half of the middle plane is shown. Without EMBr, the predicted TKE inside the nozzle is ~0.15 m$^2$/s$^2$, while turning on just the bottom coil reduces the TKE to ~0.1 m$^2$/s$^2$. After turning on the top coil as well, the predicted TKE inside the nozzle drops to ~0.04 m$^2$/s$^2$. EMBr dramatically suppresses TKE in the SEN bottom and port exit as well. The high turbulence region around the jet becomes thinner and shorter when EMBr is used. Near the top surface, turning on the bottom EMBr ruler causes the TKE to drop from ~0.06 m$^2$/s$^2$ to 0.
\[ \sim 0.005 \text{ m}^2/\text{s}^2. \] Adding the top ruler further reduces the TKE to 0.0005 m\(^2\)/s\(^2\).

![Image](image)

**Figure 5.31:** Predicted TKE at the middle plane for cases with submergence depth 170 mm and varies magnetic field strengths a) without EMBr (b) bottom coil current 850 A; (c) top coil 400 A and bottom coil 850 A; (d) both coils 850 A

### 5.6 Summary and Conclusions

A multi-GPU-based LES code CUFLOW is applied to study the effect of submergence depth and double-ruler EMBr on steel flow in the nozzle and mold region of a typical commercial steel caster. The important findings of this work are summarized below:

1. EMBr causes flow in the nozzle to be more uniform, reducing the extent of asymmetric flow caused by the slide gate. It also increases downward velocity and momentum along the SEN walls, forming an M-shaped velocity profile in the lower part of the SEN.

2. Swirl in the nozzle bottom decreases with increasing casting speed (via the accompanying increase in slide gate opening) and with increasing EMBr field strength across the nozzle.

3. Back-flow is often seen in the top portion of the port, or in the core of the big swirl exiting the port. Increasing casting speed from 1.3 m/min to 1.8 m/min has little effect on the size of the back flow region. EMBr makes the upper back-flow region larger (occupying more than 1/3 of the port).

4. With EMBr, the jets leaving the ports become thinner and stronger. This is affected by the M-shaped profile inside the SEN and the swirl in the nozzle bottom.

5. When the slide gate moves towards the inner radius side of the nozzle, the time averaged velocity near the outer radius is higher, although the flow direction is straight towards the SEN without cross-flow. Close to the SEN, although the mean velocity is small (<0.1 m/s), the transient results have high velocity (\(\sim 0.4\) m/s) regions close to the nozzle. With EMBr, the top surface velocity is more uniform with no velocity bias to either side.
6. Without EMBr, vortices form in the corner region close to the NF wall which rotate opposite to the main top recirculation region. These vortices are suppressed with EMBr.

7. Increasing casting speed from 1.5 to 1.8 m/min without EMBr, causes the top surface velocity to exceed 0.4 m/s, the surface profile variations to exceed 10 mm, and surface level fluctuations of ∼13 mm, which may cause slag entrainment. These variations are all lowered with EMBr.

8. However, with both EMBr rulers operating in the caster of this study, holding the meniscus level at middle of the top ruler of the EMBr greatly reduces the top surface velocity (to ∼0.05 m/s), which may cause inadequate heat transfer at meniscus and may lead to meniscus freezing and slag entrapment.

9. EMBr modifies the top surface level profile from a “W” shape to a “U” shape, lessens its variations, and reduces the turbulence kinetic energy at the top surface.

10. For best steel quality, it is recommended to operate this caster at 1.8 m/min with only the bottom EMBr ruler turned on at 850 A.

5.7 References


Chapter 6

Single Argon Bubble Rise in Liquid Steel under External Magnetic Field

This chapter presents a fundamental study of effects of transverse magnetic field on the motion of bubbles rising in a electrically conducting liquid. A VOF approach is used to fully resolve the interface of the argon bubble. The motion and terminal velocities of the rising bubble under different magnetic fields are compared and a reduction in rise velocity is seen in cases with the magnetic field applied. The shape deformation and the path of the bubble are discussed. An elongation of the bubble along the field direction is seen, and the physics behind these phenomena is discussed. A modified drag coefficient is obtained to include the additional resistance force caused by adding transverse magnetic field. The modified drag coefficient is then used in the Eulerian-Lagrangian model (see Chapter 7) to study the argon bubble transport in a commercial caster. The model and results presented in this chapter has been submitted to the Physics of Fluids for consideration of publication[1].

6.1 Introduction

In metallurgical processes, in order to mix and homogenize the metal, gas bubbles are injected at the bottom of a bulk liquid metal to stir the liquid metal [2]. In the process for continuous casting of steel, argon bubbles are commonly injected during the casting process. Understanding the motion of such argon bubbles is important as it has been shown that inclusions can be removed by bubble flotation [3]. In addition, in order to improve the product quality frequently an external magnetic field is applied to control the fluid motion and bubble behavior. In the past several decades, numerous [4–12] theoretical [6], experimental [7–9] and computational studies [9–12] have been carried out on the dynamics of a rising bubble in transparent liquids (such as water and oils). However, only a limited number studies have been reported on bubble motion in liquids when subjected to an external magnetic field. Since these liquids are usually metals and thus opaque, experiments to measure the bubble velocity have been difficult.

There are several important dimensionless numbers that govern the dynamics of the bubble rise. In most cases the Reynolds number is defined as $Re_b = \frac{\rho l \sqrt{gd} d \mu_l^{-1}}{}$ where $d$ is the bubble diameter, $\rho$ is
density, \( \mu \) is viscosity, \( g \) is the standard acceleration due to gravity. The subscripts \( l \) and \( g \) denote liquid and gas, respectively. The characteristic velocity used is \( \sqrt{gd} \) and sometimes it is also replaced by the bubble terminal rising velocity \( u_\tau \), which yields the terminal Reynolds number \( Re_\tau = \rho_l u_\tau d \mu_l^{-1} \). The Eötvös number \( Eo = (\rho_l - \rho_g) gd^2\gamma^{-1} \) (when \( \rho_l \gg \rho_g \), the Eötvös number can be approximated by the Bond number \( Bo = \rho_l gd^2\gamma^{-1} \), where \( \gamma \) denotes the surface tension) reflects the importance of surface tension force to gravitational force. It is used together with Morton number \( Mo = g\mu_l^4 (\rho_l - \rho_g) \rho_l^{-2}\gamma^{-3} \) to characterize the shape of bubbles. For a given pair of liquid and gas, \( Mo \) is fixed and \( Bo \) depends on the bubble size.

The Archimedes number \( Ar = \rho_l |\rho_l - \rho_g| gd^3\mu_l^{-2} \) gives the ratio of gravitational force to the viscous force. The confinement ratio is defined as \( Cr = W/d \), where \( W \) is the width of the duct. The confinement ratio affects both the rise velocity and the path of the bubble.[9] \( N \) is Stuart number (also known as magnetic interaction parameter) that describes the ratio of electromagnetic to inertial forces and \( N = Ha^2 Re_b^{-1} \), where the Hartmann number \( Ha \) is defined as \( Ha = Bd\sqrt{(\sigma/\mu)} \), \( B \) is the strength of the magnetic field and \( \sigma \) is the electrical conductivity.

Table 6.1 lists previous publications that have investigated the motion of single bubble rise in a conducting liquid with an external magnetic field. The terminal velocity of the bubble were measured by either the electrical triple probe method [13] or the Ultrasound Doppler Velocimetry method (UDV) [14, 15].

<table>
<thead>
<tr>
<th>Year</th>
<th>Author</th>
<th>E/C*</th>
<th>Methods</th>
<th>Material</th>
<th>Field Direction</th>
<th>Field Strength</th>
<th>Bubble Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>1997</td>
<td>Y. Mori, et al</td>
<td>E</td>
<td>Electrical triple probe</td>
<td>N(_2) and Mercury</td>
<td>horizontal</td>
<td>B = 0 - 1.5 T, d = 1.8 - 5.8 mm</td>
<td></td>
</tr>
<tr>
<td>2005</td>
<td>C. Zhang, et al</td>
<td>E</td>
<td>UDV</td>
<td>Ar and GaInSn</td>
<td></td>
<td>B = 0 - 0.3 T, N = 0 - 2, d = 4.5 - 8.5 mm, Eo = 2.2 - 6.6</td>
<td></td>
</tr>
<tr>
<td>2014</td>
<td>S. Schwarz, et al</td>
<td>C</td>
<td>Euler-Lagrangian + IBM</td>
<td>Ar and GaInSn</td>
<td>vertical</td>
<td>N = 0 - 2, d = 3 - 4.6 mm, Eo = 1.05 - 2.5</td>
<td></td>
</tr>
<tr>
<td>2014</td>
<td>J. Zhang, et al</td>
<td>C</td>
<td>VOF + AMR</td>
<td>Ar and GaInSn</td>
<td></td>
<td>B = 0 - 0.5 T, N = 0, 4.3, 65, d = 2.5 - 6.5 mm, Eo = 0.74 - 4.9</td>
<td></td>
</tr>
<tr>
<td>2010</td>
<td>Y. Shibasaki, et al</td>
<td>C</td>
<td>VOF + HS-MAC</td>
<td>N(_2) and Mercury</td>
<td>horizontal</td>
<td>Ha = 0, 25, 100, 200, Eo = 4.98</td>
<td></td>
</tr>
<tr>
<td>2014</td>
<td>J. Zhang, et al</td>
<td>C</td>
<td>VOF + AMR</td>
<td>Ar and GaInSn</td>
<td>horizontal</td>
<td>N = 0, 25, 100, Eo = 24.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Ar and GaInSn</td>
<td>vertical</td>
<td>B = 0, 0.3, 0.9, 1.5 T, d = 3 - 4.5 mm, Eo = 1.2 - 2.5</td>
<td></td>
</tr>
</tbody>
</table>

*E – Experimental; C – Computational;

One of the early experimental works which studied bubbles rising in a conducting metal is by Mori et al. [13] which reported experimental results of a nitrogen bubble rising in mercury with and without an external
magnetic field. They used an electrical triple probe to measure the rise velocity and the shape of the rising bubble. A cylindrical container of 500 mm in height and 75 mm inner diameter was filled with mercury to ~300 mm in height, and a magnetic field with strength $B$ between 0 ~ 1.5 T was applied in the transverse direction. Bubbles were then injected at the bottom of the container with sizes varying from 0.1 to 3 mm. Their study showed that the effect of a magnetic field on bubble rise velocity depends on the bubble radius. For small bubbles (radius around 1 mm) a temporary increase of rise velocity was seen, but for larger bubbles (radius around 3 mm) the results showed that the terminal velocity decreases with increasing magnetic field strength. Small bubbles (radius around 1 mm) rose almost in a rectilinear path when the magnetic field was applied but rose spirally without the magnetic field. The suppression of spiral motion by the magnetic field increased the rise velocity.

Zhang et al. [14] used UDV to study a single argon gas bubble rising in a cylindrical container (diameter of 100 mm) filled with GaInSn (an eutectic alloy) up to 220 mm under the influence of an external DC longitudinal magnetic field. Different strengths of the magnetic field were applied with the maximum magnetic field around 0.3T, corresponding to a magnetic interaction parameter $N = Ha^2Re^{-1}$ close to unity ($Ha$ and $Re$ are Hartmann number and Reynolds number, respectively). The effect of the magnetic field on bubbles of different size was also studied with diameters ranging from 4.5 mm to 8.5 mm. Their measurements indicated that both vertical and radial velocity components of the liquid were considerably modified by applying the magnetic field and the oscillations in the bubble wake were suppressed. They observed that with a magnetic field of 0.17 T the amplitude of the radial velocity in the wake was five times smaller than that without the magnetic field. The electromagnetic field damped the liquid velocity, leading to a more rectilinear bubble trajectory, consistent with previous observation [13]. The drag coefficient was extrapolated from the measurements. It was seen that for bubbles with diameter less than 4.6 mm, an increase in magnetic interaction parameter $N$ leads to an increase of the drag coefficient. However for larger bubbles (diameter larger than 5.4 mm) the application of a magnetic field reduces the drag coefficient. Because of the limitations in the experimental capability, they were not able to measure the bubble shape.

Recently, Schwarz and Fröhlich [16] numerically simulated the flow configuration of Zhang et al. [14]. In this work, the motion of a single argon bubble rising in a square duct with an external longitudinal magnetic field was simulated using an in-house code “PRIME”. An Euler-Lagrangian approach was used in which the incompressible Navier-Stokes equations for the liquid metal were solved on a staggered Cartesian grid with a second order accurate finite volume method. A no-slip boundary between the bubble and the liquid was imposed. The Lorentz force was obtained by solving equations for the electric potential and the current. Shape of the argon (Ar) bubble was assumed to be an oblate ellipsoid and was represented by a Immersed
Boundary Method (IBM) [17]. The surface of bubble was described by Lagrangian marker points and the motion of the bubble was obtained by solving the linear and angular momentum equations. The aspect ratio of the bubble was prescribed as a function of the Weber number \( W e = \rho u_p^2 d / \sigma \), where \( u_p \) denotes the bubble speed. The electrical conductivity of the Ar bubble was taken to be equal to the liquid metal value as the focus of their work was on the influence of the magnetic field on the bubble wake. Their results showed a zig-zag motion of bubbles. For large bubbles increasing the magnetic interaction parameter \( N \) first caused the time-averaged bubble rise velocity to increase and then decrease, but for small bubbles an increase in the magnetic field caused a decrease in time-averaged bubble rise velocity. They also showed that with increasing \( N \), the amplitude of the oscillation of the bubble path, the dimensionless characteristic frequency of oscillations and the Strouhal number decrease.

To study the bubble shape in the experiment done by Zhang et al. [14], several Direct Numerical Simulations (DNS) were carried out by Zhang and Ni [18]. A Volume of Fluid (VOF) method was applied to obtain the shape of the bubble and an Adaptive Mesh Refinement (AMR) technique was used to better capture the gas-liquid interface. Equations for the magnetic potential and the current density were solved to calculate the induced Lorentz force on fluid. The bubble diameter \( d \) varied from 2.5 mm to 6.5 mm, Eötvös number \( Eo \approx 0.74 \sim 4.9 \) and Morton number \( Mo = 2.4 \times 10^{-13} \) with a maximum magnetic field of 0.3 T. The bubble Reynolds number based on bubble terminal velocity ranged from 2000 to 4000. A spatial resolution of \( \Delta x = 0.02d \) was used across the interface. Their results showed that a moderate magnetic field yields an increased terminal velocity, while a stronger magnetic field leads to a reduced terminal velocity irrespective of the bubble size, which is different from experimental results of Zhang et al. [14]. They also observed a significant difference in terminal velocity when compared with measurements reported by Zhang et al. [14]. When the strength of the magnetic field was increased, a second path instability was seen. The wake structure behind the bubble was more regular and parallel when the magnetic field was applied.

Using the UDV, Zhang et al. [19] investigated the flow structure of bubble motion in a liquid metal under the presence of a horizontal magnetic field. In this work, a cylindrical container with diameter of 90 mm and height of 220 mm was filled with GaInSn, and gas was injected through a single-hole nozzle (inner diameter of 1 mm) placed at the bottom of the container. A transverse DC magnetic field was applied which could yield a maximum Hartmann number \( (Ha) \) of 484. By studying the velocity field with Hartmann numbers of 0, 162, 271 and 484, they showed that in the plane parallel to the magnetic field lines, a suppression of descending flow and a promotion of ascending flow occurs with increasing \( Ha \). However, the opposite behavior was found in the perpendicular plane. Most cases were studied with the interaction parameter \( N \) ranging from 1 \( (Ha = 162) \) to 10 \( (Ha = 484) \). It was seen that the presence of a moderate magnetic field
destabilizes the global flow and leads to transient, oscillating flow patterns with predominant frequencies. However, by using a very small gas flow rate, $N$ was increased to around 50 and global damping of the flow was observed. Their results also showed that the root-mean-square (RMS) value of the vertical velocity increases with moderate Hartmann numbers ($Ha < 400$). They noticed that the concentration of bubbles becomes more non-isotropic and more pronounced along the direction parallel to the magnetic field.

In a subsequent experimental work, Zhang et al. [15] further studied the damping effect of transverse and longitudinal magnetic fields. Similar to the previous study, a cylindrical container of 90 mm diameter and 220 mm height was filled with GaInSn and Ar bubbles were injected from a 1 mm nozzle placed at the center and bottom of the container. In this work, a fixed gas flow rate $Q = 0.33 \text{ cm}^3/\text{s}$ was used to generate small and separated bubbles. A maximum magnetic field strength of 0.28 T corresponding to a Hartmann number of 484 was used. The measurements showed that the direction of the magnetic field influences the flow structure in the container. With the presence of a longitudinal magnetic field, a global damping of the flow was observed but with a horizontal magnetic field the flow pattern was quite different. Their results showed that with the horizontal magnetic field, the measured velocity was no longer axisymmetric. The liquid recirculation was intensified in the plane perpendicular to the magnetic field, but was suppressed in the plane parallel. The original circular jet was seen to be stretched along its cross section parallel to the magnetic field direction, and those vortices whose axes are in line with the magnetic field direction were only weakly damped.

To simulate the above experimental studies, Miao et al. [20] used a commercial software package CFX to study the bubble driven liquid metal flow in the presence of horizontal and longitudinal magnetic fields. An Eulerian-Eulerian approach with modified RANS-SST model was used to model the GaInSn liquid and the gas phase. Based on different gas flow rates, the bubble diameters used were 4.4, 5.5 and 8.2 mm. The breakup and coalescence of bubbles were neglected in their work. They proposed a modified turbulence model by adding an anisotropy variable $\alpha_\mu$, and sink terms (function of $\alpha_\mu$) for $k$ and $\epsilon$ were added to include the effect of magnetic field on turbulence. They used both a conventional turbulence model and the modified model with anisotropy variable, and their results showed that the modified model agreed with measurements better than the common RANS model. With a transverse magnetic field, the simulations showed that the vortex was elongated along the direction of the magnetic field and after applying the magnetic field the dispersion of bubbles was suppressed and concentrated in the core region of the container. An anisotropic distribution of bubbles was seen with $Ha = 271$. After increasing the Hartmann number to moderate values, the total kinetic energy increased, but further increase of the Hartmann number gradually decreased the total momentum because the flow was dominated by the damping effect of the Lorentz force.
Tagawa [21] used a finite difference method with Highly Simplified Marker and Cell (HSMAC) algorithm to study the dynamics of a falling droplet of liquid metal into liquid metal pool and an air bubble rising in water subject to a magnetic field. For the later problem, a cylindrical container with diameter $2d$ and height $8d$ was used. At the middle height of the enclosure, an electromagnetic coil with radius $2d$ was placed to impose an axisymmetric, non-uniform magnetic field (calculated with the Biot-Savart’s law and has a maximum $B$ of 1.7 T). The Lorentz force was only in the radial direction and was computed explicitly. The liquid, gas and the finite thickness interface were solved simultaneously with a index function $\phi$ to locate the interface. The governing equations in an axisymmetric coordinate system were solved using $128 \times 1024$ grid points. The results showed that at the beginning the bubble deformed quickly but kept the same shape until it reached the magnetic field region. The bubble was then seen to be elongated horizontally at the middle of the enclosure. Since an axisymmetric assumption was used in the simulation, no zig-zag motion or spiral motion of the bubble was observed.

Korlie et al. [22] developed a VOF code for DNS of two-phase flow with magnetic fluids. They studied the motion of a non-magnetic 2D bubble rising in a ferro-fluid, and a droplet of ferro-fluid falling through a non-magnetic medium with a magnetic field imposed. Equilibrium magnetization and linear magnetic material were prescribed. The simulations were performed with a grid of $128 \times 128$ nodes. The Bond number was kept close to 1. Their results showed that for a bubble with diameter of 1.5 mm, the magnetic field makes the bubble elongate in the rising direction, leading to a drag reduction which in turn leads to a faster rise velocity. Breakup of the bubble was seen when the magnetic force was large and the magnetic susceptibility $\chi$ exceeded 4.

Related to the steel making process, Haverkort and Peeters [23] studied the effects of magnetic field on insulating bubbles and inclusions in the continuous casting process. Several steady state simulations were carried out to study the Ar gas bubble behavior in steel under a magnetic field applied perpendicular to the flow. To include the effect of magnetic field on the drag of the spherical bubble, an experimental correlation of $C_D = C_{D0} \left(1 + 0.7\sqrt{N}\right)$ was implemented, where $C_{D0}$ is the drag coefficient in the absence of magnetic field and $N$ is magnetic interaction parameter. Their results showed that even if the particles/bubbles were non-conducting, the magnetic field can affect their motion through modifying the behavior of surrounding fluid. The terminal rise velocity of bubbles was observed to be fairly insensitive to $N$ but was affected by $Ha$ and $Re$. The drag coefficient of a rigid sphere moving perpendicular to the magnetic field increased approximately proportional to the square root of $N$ (agreed with the formulation of drag coefficient that was implemented). Their results also showed that the distribution of gas near the side walls of the submerged entry nozzle was affected by the magnetic field.
Shibasaki et al. [24] also studied the rise of a single bubble in a square duct subject to a vertical magnetic field. A square duct with cross section of $2d \times 2d$, and a height of $6d$ was used. Simulations were carried out on a grid containing $60 \times 60 \times 180$ grid points. The Hartmann number $Ha$ equal to 0, 25, 50, 75, 100 and 200 were used. The Galilei number $(gL^3\nu^{-2})$ used in their study was $4.0 \times 10^4$. They observed that with increase of the magnetic field strength, the shape of the bubble elongated in the rising direction. Low pressure regions in the upper and lower parts of the bubble were observed when magnetic field was strong, which was attributed to cause the elongation of the bubble. The results also showed that with an increase in the strength of the magnetic field, the vortices behind the bubble tend to disappear due to the strong braking force in the transverse direction of the flow. The results also showed that the rise velocity of the bubble for $Ha < 75$ is slightly larger than that without the magnetic field, but for $Ha > 75$ the predicted rise velocities decrease monotonically with increasing $Ha$. They also observed that in the region of $0 < Ha < 75$ the flow was quite complex, and the relationship of the Hartmann number and the drag reduction were not clearly understood.

Liu and Pan [25] used the VOF method to study a single bubble rising and bubble coalescence in a cylindrical container (20 mm in diameter and 50 mm in height) filled with conducting liquid (electrical conductivity 100 S/m) with an uniform external magnetic field of 2 Tesla applied in the axial direction of the container. Surface tension effect was modeled by the Continuum Surface Force (CSF) method. A spherical bubble of diameter about 5 mm was initially placed at the bottom of the container. By examining the bubble shape at time of 0.03 s, 0.09 s and 0.18 s for the cases with and without a magnetic field, they observed that when no magnetic field was applied the bubble deformed quickly into a spherical cap and stayed as that for rest of the time. When the magnetic field was applied, the bubble shape was seen to be less squeezed in the rising direction and the bubble was less deformed. No appreciable increase of rise velocity was seen.

Zhang and Ni [26] further went on to study the motion of a single gas bubble rising in a square duct filled with an electrically conducting liquid under a transverse magnetic field. For their system, a bubble Reynolds number $Re_b = 125$, Weber number $We = 6.5$ and the Eötvös number $Eo = 24.5$ were chosen. The minimum size of the grid was $\Delta x = d/64$. The magnetic interaction parameter $N$ was set to 0, 25 and 100. Their results showed that the computed bubble shape without a magnetic field was symmetrical about vertical and horizontal diameters but was elliptic under the influence of the magnetic field. Under the strong magnetic field ($N = 25$ and 100) the bubble was greatly compressed in the direction parallel to the magnetic field. This result is in contradiction with the works of Zhang and Ni [18], Shibasaki et al. [24], Shin and Kang [27] and Zhang and Ni [18], where the bubble is elongated in the direction of magnetic field.
terminal velocity was seen to decrease and become less oscillatory with the application of the transverse magnetic field. The streamlines in the plane parallel to the magnetic field and normal to the magnetic field were observed to be different.

Summarizing the key publications in literature, an external magnetic field is seen to make the bubble rise slower [13, 14, 16, 18, 24, 26] with a more rectilinear rising path [13, 14, 16], with suppression and stabilization of the wake behind the bubble [14, 16, 18, 24]. However to date, most of the works considered a vertical magnetic field [14, 16, 18, 24, 28, 29] with only two studies with a horizontal magnetic field configuration. [13, 26] The effect of a horizontal magnetic field on the bubble shape is still not well understood. Further, to our knowledge there are no studies that have examined the case of an argon bubble rising in liquid steel with a magnetic field. In this work, we study the effects of a transverse magnetic field on an argon bubble rising through a column of molten steel. We present a number of results from our simulations including the terminal velocity of the rising bubble, the wake structures and the bubble path. The mechanisms of bubble elongation along field direction are discussed. From the results of this study, a modified drag coefficient is extracted to be used in practical calculations with large number of isolated bubbles (Lagrangian description).

This chapter is organized first describing in Section 6.2 the governing equations, numerical method and solution procedure. Section 6.3 provides results of validation and grid independence studies. Subsequently, results of several well-resolved calculations are presented and discussed in Section 6.4. Finally, Section 6.5 concludes the chapter with a summary of the important results.

6.2 Governing Equations and Solution Procedure

The problem considered in this study is a single Ar bubble rising in an initially quiescent column of liquid steel with a transverse external magnetic field (parallel to $x$ axis) applied to the entire domain as shown in Fig. 6.1. The governing continuity and momentum equations are given by Eqn. (6.1) and Eqn. (6.2).

$$\nabla \cdot (\rho \mathbf{u}) = 0$$ (6.1)

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot [\mu (\nabla \mathbf{u} + \nabla \mathbf{u}^T)] + \mathbf{F}_L + \mathbf{F}_S + \rho g$$ (6.2)

where $\mathbf{u}$ is fluid’s velocity, $t$ is time, $\rho$ is fluid’s density, $\mu$ is fluid’s dynamic viscosity, $p$ is total pressure and $g$ is gravity. The two source terms $\mathbf{F}_L$ and $\mathbf{F}_S$ represent Lorentz force and surface tension force, respectively. The outer product $\mathbf{u} \otimes \mathbf{u}$ is equivalent to a matrix multiplication $\mathbf{u} \times \mathbf{u}^T$. In this work, for a Ar bubble
Figure 6.1: Computational domain and the initial bubble location

rising in liquid steel the magnetic Reynolds number $Re_m = \mu_e \sigma du/\tau$ is very small ($\sim 10^{-3}$), where $\mu_e$ is the magnetic permeability of free space. Hence, the induced magnetic field by the fluid motion is much smaller than the applied magnetic field, and therefore a quasi-static approximation can be used [30]. A potential method can then be used to compute the electric current and Lorentz force distribution. The Lorentz force $F_L$ is obtained by taking cross product of current density $J$ and external magnetic field $B$, as shown in Eqn. (6.3). The electrical current density $J$ can be computed through the Ohm’s law as given by Eqn. (6.4), and for a well conducting material the current conservation law is given by Eqn. (6.5). Therefore, the electric potential $\Phi$ satisfies Eqn. (6.6). It is to be noted that the electrical conductivity of gas and liquid have different values (the conductivity of liquid steel is $\sim 10^{20}$ larger than that of argon gas) and therefore the electrical conductivity $\sigma$ in Eqn. (6.6) cannot be canceled. With a insulated wall, the boundary condition for $\Phi$ is $\partial \Phi / \partial n = 0$.

$$F_L = J \times B$$  \hspace{1cm} (6.3)

$$J = \sigma (-\nabla \Phi + u \times B)$$  \hspace{1cm} (6.4)

$$\nabla \cdot J = 0$$  \hspace{1cm} (6.5)
\[ \nabla \cdot (\sigma \nabla \Phi) = \nabla \cdot [\sigma (u \times B)] \quad (6.6) \]

The surface tension force \( F_S \), is evaluated using Eqn. (6.7).

\[ F_S = \int_{\Gamma} \gamma \kappa n \delta (x - x_f) \, ds \quad (6.7) \]

where \( \gamma \) and \( \kappa \) denote the surface tension and mean interface curvature, respectively. \( \Gamma \) represents the interface, \( n \) denotes the normal vector of the interface, \( \delta \) is the Dirac delta function. \( x \) and \( x_f \) denote the coordinates of the cell and the interface, respectively. To capture a sharp interface and reduce spurious velocities, a Sharp Surface Force (SSF) method for modeling of the surface tension force is adapted. This SSF method, also known as Pressure Boundary Method (PBM) or Ghost Fluid Method (GFM), has been presented and discussed in detail elsewhere\[31–34\]. In this method, the surface tension in Eqn. (6.7) is treated as a pressure gradient \( -\nabla \tilde{p} \) which exactly balances the surface tension force \( F_S \) generated due to presence of the interface. By considering that this new pressure field cannot generate velocity in a static case, another pressure Poisson equation can be obtained as given by Eqn. (6.8) below.

\[ \nabla \cdot \left( \frac{\nabla \tilde{p}}{\rho} \right) = F_x^+ - F_x^- + F_y^+ - F_y^- + F_z^+ - F_z^- \quad (6.8) \]

where the six terms on the RHS are scalars and defined as

\[ F_x^+ = -\left[ \frac{\gamma \kappa}{\rho \Delta x^2} \right]_{(i+1/2,j,k)} \quad \text{and} \quad F_x^- = -\left[ \frac{\gamma \kappa}{\rho \Delta x^2} \right]_{(i-1/2,j,k)} \quad (6.9) \]

etc. The surface tension force at the interface is thus treated as a jump condition for calculating this new pressure field. The interface is tracked using the VOF method in which an evolution equation for the liquid volume fraction \( \alpha \), given by Eqn. (6.10) is solved.

\[ \frac{\partial \alpha}{\partial t} + u \cdot \nabla \alpha = 0 \quad (6.10) \]

A physical property \( \theta \) (i.e. \( \rho \), \( \mu \) and \( \sigma \)) at a given point in the domain is evaluated by linear interpolation as \( \theta = \alpha \theta_1 + (1 - \alpha) \theta_g \), where the subscript “l” denotes the property of surrounding liquid and subscript “g” denotes the property of the gas phase. In the argon-steel system, the density ratio is of the order of \( \sim 10^4 \), viscosity ratio is \( \sim 10^2 \) and electrical conductivity ratio is \( \sim 10^{20} \).
The above equations are converted to dimensionless form by using the following dimensionless variables:

\[
\begin{align*}
  x^* &= \frac{x}{d}; & u^* &= \frac{u}{\sqrt{gd}}; & t^* &= t\sqrt{\frac{g}{d}}; & \rho^* &= \frac{\rho}{\rho_l}; & \mu^* &= \frac{\mu}{\mu_l}; \\
  p^* &= \frac{p}{\rho_lgd}; & \kappa^* &= \frac{\kappa}{d}; & g^* &= \frac{g}{g}; & B^* &= \frac{B}{B_0}; & \Phi^* &= \frac{\Phi}{B_0d\sqrt{gd}}; & \sigma^* &= \frac{\sigma}{\sigma_l}.
\end{align*}
\] (6.11)

The momentum equation is then re-written in a dimensionless form:

\[
\frac{\partial \rho^* u^*}{\partial t^*} + \nabla \cdot (\rho^* u^* u^*) = -\nabla p^* + \frac{1}{\sqrt{Ar}} \nabla \cdot \left[ \mu^* \left( \nabla u^* + \nabla u^*^T \right) \right] + \frac{Ha^2}{Re_b} F_L^* + \frac{1}{Bo} F_S^* + \rho^* g^*
\] (6.12)

A general purpose in-house code, CUFLOW [34–38] for simulating laminar and turbulent flows was used to solve the above equations. The code employs Cartesian grids to integrate the three-dimensional unsteady incompressible Navier-Stokes equations. The continuity and momentum equations are solved using a fractional step method. Figure 6.2 shows the solution steps at each timestep. Details of the solution algorithm and numerical implementation are available in references [34–38]. The most computationally intensive parts are the solutions of the three Poisson equations (pressure-Poisson equation, electrical-Poisson equation and the surface tension related Poisson equation). In this work, the three Poisson equations are solved efficiently by a V-cycle multigrid method, and red-black Successive Over Relaxation (SOR) with over-relaxation parameter of 1.6.
6.3 Code Validation and Grid Independence Study

The solver has been previously validated in a number of problems such as the lid-driven cavity flow of a Newtonian fluid with and without a magnetic field against published results.[38] The VOF with SSF implementations were also validated through comparing predicted bubble rise velocity and shape at high Morton numbers.[34] To further validate the algorithm in a low Morton number and high density ratio regime, two more validations are performed: 1) prediction of the shape and rise velocities of single air bubbles rising in water (tested 6 different bubble sizes); and 2) the rise velocity of a argon bubble rising in GaInSn.

6.3.1 Validation 1. - Rise of an air bubble in water

Several simulations of air bubbles of different sizes rising in a container with quiescent water were first carried and compared with previous experimental and computational works. The properties of air and water used in these validations are listed in Table 6.2. The important dimensionless numbers are provided in Table 6.3. It’s important to note that both systems (Ar-steel and air-water) are in the low Eötvös number and low Morton number regime, and therefore yield similar bubble Reynolds numbers (around 550 for a 3 mm bubble).

<table>
<thead>
<tr>
<th></th>
<th>Air</th>
<th>Water</th>
<th>Argon</th>
<th>Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T$ (K)</td>
<td>300</td>
<td></td>
<td>1773</td>
<td></td>
</tr>
<tr>
<td>$\gamma$ (N/m)</td>
<td>0.0712</td>
<td></td>
<td>1.2</td>
<td></td>
</tr>
<tr>
<td>$\rho$ (kg/m$^3$)</td>
<td>1.17</td>
<td>1000</td>
<td>0.56</td>
<td>7000</td>
</tr>
<tr>
<td>$\mu$ (kg/(m·s))</td>
<td>$1.86 \times 10^{-5}$</td>
<td>0.001</td>
<td>$7.42 \times 10^{-5}$</td>
<td>0.0063</td>
</tr>
<tr>
<td>$\sigma$ (1/(Ω·s))</td>
<td>$1.00 \times 10^{-15}$</td>
<td>0.001</td>
<td>$1.00 \times 10^{-15}$</td>
<td>714000</td>
</tr>
</tbody>
</table>

6.3. It’s important to note that both systems (Ar-steel and air-water) are in the low Eötvös number and low Morton number regime, and therefore yield similar bubble Reynolds numbers (around 550 for a 3 mm bubble).

<table>
<thead>
<tr>
<th></th>
<th>Air-water</th>
<th>Ar-steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_o$</td>
<td>1.24</td>
<td>0.51</td>
</tr>
<tr>
<td>$M_o$</td>
<td>$2.7 \times 10^{-11}$</td>
<td>$1.3 \times 10^{-12}$</td>
</tr>
<tr>
<td>$Re_b$</td>
<td>514.4</td>
<td>571.5</td>
</tr>
<tr>
<td>$\rho_l/\rho_g$</td>
<td>$8.547 \times 10^4$</td>
<td>$1.250 \times 10^4$</td>
</tr>
<tr>
<td>$\mu_l/\mu_g$</td>
<td>53.8</td>
<td>84.3</td>
</tr>
<tr>
<td>$\sigma_l/\sigma_g$</td>
<td>$1.0 \times 10^{12}$</td>
<td>$7.4 \times 10^{20}$</td>
</tr>
</tbody>
</table>

Six validation simulations were conducted for air bubbles rising in water. The selected air bubbles have diameters ranging from 1 mm to 7 mm which correspond to Eötvös numbers from 0.1 to 6.7. In each case, a single bubble was released from the bottom of a container (4$d$ in width and 10$d$ in bubble rising direction). The shape of bubble was initialized to be a sphere with zero velocity and was placed at 1$d$ above the bottom wall of the container. For bubbles with different Eötvös number, the predicted terminal bubble Reynolds numbers $Re_r$ are plotted in Fig. 6.3. The bubble shape and iso-Morton number lines are obtained from
previous studies\cite{39, 40}. It can be seen that all the points lie between the two lines of $Mo = 10^{-10}$ and $10^{-12}$ (if the $Mo = 10^{-12}$ is extended) and those points would be roughly close to the line of $Mo = 10^{-11}$ which is the Morton number for air and water. It’s important to mention that the observed bubble shapes also agree well with the Grace diagram: the 1 mm bubble was found to be spherical shape, and the 7 mm bubble was seen to oscillate between an oblate spherical and a “disk” shape. All other bubbles were found to be oblate spherical shape.

The terminal velocity of the bubble is plotted in Fig. 6.4(a) together with some of the available data. In Fig. 6.4a the upper dashed line is the terminal velocity of an air bubble rising in distilled water\cite{4} based on previous experimental work\cite{8}, while the bottom line is the terminal velocity of an air bubble rising in surfactant added water. One can see that with surfactants added, the terminal rise velocity is reduced by as much as $\sim 60\%$. It’s important to mention that these two lines merge for bubbles of small size because even a small amount of surfactant contained in the distilled water can prevent circulation inside of the bubble \cite{4}, leading to higher drag and lower terminal velocity. The triangles are UDV measurements.\cite{14} The predicted terminal velocities using our current code are close to the 3D Front Tracking Method (FTM) simulation results reported by Dijkhuizen et al. \cite{10} It can be seen from the figure that both numerical methods predict slightly higher velocities than observed in UDV experiments. The smaller terminal velocities in the experiments may be due to the small amounts of contamination still contained in distilled water which affects the circulation inside the bubble and breaks the “boundary condition” between the air bubble and the surrounding water. This leads to a higher drag and a lower terminal velocity. To minimize the impurities,
Wu and Gharib [41] performed experiments of a single air bubble (up to ~2 mm) rising in water using filtered air (0.2 µm air filters) and clean water which is taken from a deionized water source that has been pretreated by a water purification system, then distilled by an auto-distiller and filtered by a 3-module filtration system. Bubbles were slowly injected from a capillary of diameter 0.0267cm. Their results are shown as red “+” markers in Fig. 6.4(a) which match very well with our predictions. Fig. 6.4(b) compares the shape of a 2.0 mm bubble from the experiments of Wu and Gharib [41] after it travels ~6.7 mm and present computations. Both the experiment and the present simulation show similar ellipsoidal bubble shapes. Dijkhuizen et al. [42] have also mentioned that their results match closely with the experimental work of Veldhuis [43] who studied the rise of air bubbles in “ultra-pure water”. In these experiments when the bubble size is large ($d > 1.3$ mm), a zig-zag or helical motion was reported [4, 7] which is also seen in our simulation for $d \geq 2$ mm bubbles. For $d = 1$ mm, the rectilinear bubble path seen in most experiments [4, 7] is predicted in our simulation. The terminal velocity matches very well in this case. However, it’s important to point out that the predicted terminal velocity even for the 1 mm bubble is still slightly higher than that observed in the experiment due to the fact that “pure” water used in numerical simulations yields a free mobile interface, which leads to lower drag and higher rising velocity.

6.3.2 Validation 2. - argon bubble rise in GaInSn

A second validation test and grid independence study was next done by simulating an argon bubble of $d = 2.5$ mm rising in quiescent GaInSn container ($C_r = 2$) with a height of $8d$. The bubble is initialized as
a sphere and placed at the center of the duct and 1d above the bottom wall. Three simulations using 24, 32 and 48 cells across a single bubble were performed. The properties [14, 18] of GaInSn and Ar gas at 293 K used in these three simulations are given in Table 6.4. The dimensionless numbers for these simulations are

<table>
<thead>
<tr>
<th>Property</th>
<th>Argon</th>
<th>GaInSn</th>
</tr>
</thead>
<tbody>
<tr>
<td>γ (N/m)</td>
<td>0.533</td>
<td></td>
</tr>
<tr>
<td>ρ (kg/m³)</td>
<td>$1.654 \times 10^0$</td>
<td>$6.361 \times 10^3$</td>
</tr>
<tr>
<td>μ (kg/(m·s))</td>
<td>$1.176 \times 10^{-5}$</td>
<td>$2.200 \times 10^{-3}$</td>
</tr>
</tbody>
</table>

$Bo = 0.74$, $Mo = 2.38 \times 10^{-13}$ and $Re_b = 1140$. The bubble rise history curves for the three grid resolutions are plotted in Fig. 6.5a. As the grid is refined, the rise velocity increases slightly. Comparing these curves, we infer that $\Delta x = d/32$ and $\Delta x = d/48$ are in close agreement. Refining the mesh from $\Delta x = d/32$ to $\Delta x = d/48$ changes the predicted terminal velocity only by 1.6%. Therefore, we have used $\Delta x = d/32$ in our simulations.

In Fig. 6.5b the predicted rise velocity of this 2.5 mm bubble is plotted against the UDV measured terminal velocity[14], other numerical prediction [18] and a theoretical prediction [6]. The line in Fig. 6.5b is the terminal velocity suggested by Mendelson [6] who used an analogy between waves and bubbles, and treated the large bubbles as an interfacial disturbance. By using $\pi d$ as the wave length, the terminal velocity (equal to the traveling speed of the wave) can be obtained as

$$u_r = \sqrt{2\gamma d^{-1} \rho^{-1} + 2gd} \quad (6.13)$$
It has been shown that this wave analogy predicts rising velocity close to experimental data in the air-water system for bubble diameter greater than 1.3 mm [4]. In Fig. 6.5b, the measured terminal velocities in the UDV experiments are always lower than those predicted by the theory, which may due to the impurities (oxides) in the GaInSn that lead to a lower surface tension [14] and less slippery interface between the bubble and the surrounding liquid. Thus the drag is increased and the rise velocity is reduced. Our numerical result is closer to that reported by Zhang and Ni [18] and the theoretical solution [6].

6.3.3 Validation 3. - N$_2$ bubble rise in mercury with horizontal magnetic field

To further validate the multi-physics code, a bubble rising in an electrically conducting fluid with a magnetic field is considered. Although measurements [14] are available for single argon bubble rising in GaInSn with a vertical magnetic field, the measured rise velocity is influenced by the contaminations (oxides) in the GaInSn. Thus, a simulation of a N$_2$ bubble ($d = 5.6$ mm) rising in mercury with a horizontal magnetic field of 0.5 T was conducted on a grid consisting of $128 \times 128 \times 384$ cells. The computational domain is the same as that shown in Figure 6.1. The physical properties of mercury and N$_2$ are given in Table 6.5. Figure 6.6 compares the predicted rise velocity with experimental results [13] and previous numerical predictions [26]. The result show that with a horizontal magnetic field of 0.5 T, a $d = 5.6$ mm N$_2$ bubble in mercury rises at

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure6_6.png}
\caption{Rise velocity of a $d = 5.6$ (Eo = 8.5) mm N$_2$ bubble rise in mercury subject to a horizontal magnetic filed of 0.5 T ($N = 0.45$) $\sim$162 mm/s, which is very close to the measured [13] rise velocity of $\sim$168 mm/s and the previous numerical prediction [26] of $\sim$175 mm/s. The measured velocity reported by Mori et al. [13] is a time-averaged velocity.

\begin{table}[h]
\centering
\begin{tabular}{lcc}
\hline
 & N$_2$ & Mercury \\
\hline
$\gamma$ (N/m) & 0.4865
\hline
$\rho$ (kg/m$^3$) & 1.17 & 1.35 $\times 10^4$
\hline
$\mu$ (kg/(m·s)) & $1.77 \times 10^{-5}$ & $1.50 \times 10^{-3}$
\hline
$\sigma$ (1/(Ω·s)) & $1.0 \times 10^{-15}$ & $1.02 \times 10^6$
\hline
\end{tabular}
\caption{Physical properties of N$_2$ and mercury at $T \approx 300$K}
\end{table}
after the bubble passes the initial rise stage. Note that, in the case without the magnetic field, the predicted mean rise velocity of the \( \text{N}_2 \) bubble is 208 mm/s with large shape and velocity oscillations. The predicted mean rise velocity agrees well with the analytical prediction of 199 mm/s (Mendelson equation [6] given by equation (6.13)) and also matches with that reported by Zhang and Ni [26].

6.4 Results and Discussions

We now present results of several numerical studies in the geometry shown in Fig. 6.1. The computational domain has a cross section of \( 6d \times 6d \) and a length of \( 16d \). A grid of \( 192 \times 192 \times 512 \) (\( \sim 19 \text{ million} \)) cells in the \( x, y \) and \( z \) direction was used to discretize the governing equations. A spherical argon bubble of diameter \( d \) was initially placed at the center and \( 0.5d \) above the bottom wall of the container. A uniform magnetic field of strength \( B \) was applied in the horizontal direction (\( x \)) and kept constant throughout the simulation. The dynamics of the bubble, surrounding fluid, and their interaction with the Lorentz force during the rise process are studied. The physical properties of the hot argon gas and the liquid steel are provided in Table 6.2. Table 6.6, lists parameters of 9 simulations with different bubble size and magnetic field strength combined in this study. The cases investigated have a \( Eo \) of 0.51 or 2.80, \( Re_b \) was varied from 572 to 2037 and \( Ha \) was increased from 0 to 37.26. The results are presented in the following structure: Section 6.4.1 focuses on the effect of the magnetic field on bubble rise velocity, deformation and oscillation frequency. Section 6.4.2 analyzes the velocity field and the Lorentz force distribution. The interaction between the velocity field, bubble shape and the Lorentz force are discussed; Section 6.4.3 presents the wake structures behind the rising bubble; Section 6.4.4 studies the effect of the magnetic field on the drag coefficient.
6.4.1 Bubble rise velocity and deformation

The dimensionless rise velocities $W^*$ versus dimensionless time $t^* = t\sqrt{g/d}$ for all cases are first shown in Fig. 6.7. The rise velocity $W^*$ is computed as the volume average of the vertical velocity $w^* = w/\sqrt{gd}$ of the bubble. For 3 mm bubbles, Fig. 6.7a shows that for all cases the rise velocity curves are smooth and non-oscillatory. In the initial stages ($t^* < 0.5$) the three rise curves are almost coincident. Without the magnetic field, the rise velocity of a 3 mm bubble first increases to a value of 2.5 and then starts to decrease slightly. This decreasing trend is caused by the inclined motion of the bubble, which will be discussed later. After applying a transverse magnetic field of $B = 0.2$ T, the rise velocity is reduced by $\sim 4\%$ from the peak value of the zero magnetic field case. In the case with zero magnetic field, the rise velocity ($w^*$) reduces after $t^* = 6$ due to a non-rectilinear motion of the bubble, however, its speed remains the highest. When a magnetic field of 0.5 T is applied, the rise velocity is seen to decrease to 1.83, a 24\% reduction from the maximum rise velocity without any magnetic field. Figure 6.7b shows the rise velocities for 5 mm bubbles. Without magnetic field, the rise velocity of a 5 mm bubble shows a similar trend as that of the 3 mm bubble. The rise velocity increases to a maximum value 1.72 and then starts to decrease. Different from the 3 mm case, the rise velocity of the 5 mm bubble slightly oscillates. After applying a magnetic field of $B = 0.22$ T ($N = 0.1$), the rise velocity is slightly reduced but still oscillatory. The oscillation completely disappears after increasing the magnetic field to 0.54 T ($N = 68$), and the rise velocity is reduced to $\sim 0.127$ which is 25\% smaller than that without magnetic field. Figure 6.7c shows the rise velocities for 7 mm bubbles. It is seen
the rise velocity for the 7 mm bubble without the magnetic field is oscillatory after the initial rise \( t^* > 1.0 \). These velocity oscillations are a result of the oscillations in bubble shape, as will be explained later. The peaks and valleys of the rise velocity correspond to smaller and larger drag on the bubble, as the bubble shape expands and contracts along different axes (the bubble volume is ensured to remain constant). With no magnetic field, the bubble oscillations persist until the end of the duct, reaching a periodic oscillatory motion. When the magnetic field of 0.2 T is applied, the amplitude of oscillations is considerably dampened. Until about \( t^* = 1.8 \), the two rise velocity curves are nearly the same, indicating nearly the same drag on the bubble and nearly same bubble deformation. However, \( t^* = 1.8 \), the rise velocity curves depart and the oscillations are dampened by the magnetic field. The amplitude decreases as a result of different bubble shapes (to be presented below) and drag. Further increase of the magnetic field to 0.5 T completely damps the oscillation, resulting in a steady rise velocity. Under the strong magnetic field of 0.5 T, the rise velocity is seen to be reduced by \( \sim 25\% \), from the no magnetic field average value. Here we focus on the dynamics of the 3 mm and 7 mm bubbles, because their shapes are in different regimes.

**Dynamics of the 3 mm bubble**

Figures 6.8 to 6.12 show top and side views of the 3 mm bubble at different times for the three magnetic field strengths. With no magnetic field, the top views of the bubble in Fig. 6.8 show that at \( t^* = 2 \) the maximum bubble cross-sectional diameter is \( \sim 1.18d \). After that, the maximum cross-sectional diameter remains the same. For \( t^* < 4 \) the bubble is almost in a rectilinear motion and remains at the center of the duct. The rise velocity plot shows that at \( t^* \approx 5 \) the rise velocity reaches its maximum and at that time it is already biased from the centerline of the duct. Subsequently, the acceleration of the bubble becomes smaller and the biased motion of the bubble causes a reduction in vertical rise velocity. At \( t^* = 6 \) the top view shows
that bubble has already tilted and it is moving toward the southeast corner of the duct. At $t^* = 6.5$ the bubble tilts further away from the centerline of the duct, further lowering the vertical component of the rise velocity.

The side and front views of the bubble for $B = 0$ T at different times are shown in Fig. 6.9. These side views clearly show that at later times, the bubble deforms to an ellipsoidal shape. A close examination of the surface curvature shows that the front of the bubble is flatter than the back of the bubble. This may be due to the flow impinging on the top and the pressure on the top surface being higher. The bubble is seen to be tilted by around 25° and is biased from the center of the duct. The dashed lines in the figure are approximate tangents to the top (outer surface) of the bubble. The average rise velocity can be estimated also using the $z$ dimension of the dashed line divided by the time interval (0.75). By using this graphical method the computed average bubble rise velocity for the 3 mm bubble for $B = 0$ T is 2.46. This estimated average rise velocity agrees well with that predicted using the volume integral method that was presented in Fig. 6.8.

The corresponding rise velocity and top views of the 3 mm bubble with the 0.2 T transverse magnetic field are shown in Fig. 6.10. With the external magnetic field, the rectilinear motion of the bubble lasts

![Figure 6.9: Bubble shape at different time $t^*$, $d = 3$ mm, $B = 0$ T](image)

![Figure 6.10: Bubble rise velocity and the associated top-view of the 3 mm bubble at different times indicated as red triangle on the line plot) with $B = 0.2$ T.](image)
longer and it is seen that at \( t^* = 5 \) the bubble still remains at the centerline of the duct. The maximum cross sectional diameter of the bubble is seen to be \( \sim 1.18d \). Top views of the bubble at \( t^* = 6 \) and 6.5 reveal that the bubble moves upwards toward the northeast corner of the duct. Although the horizontal motion of the bubble is seen for both \( B = 0.2 \) T and \( B = 0 \) T cases, the bubble is not tilted significantly. The rise velocity only slightly decreases after \( t^* = 6 \), which is again caused by the transverse motion of the bubble.

Figure 6.11 shows the side and front views of the 3 mm bubble for \( B = 0.2 \) T at different times. Comparing the bubble shape observed, it can be seen that the magnetic field of 0.2 T does not significantly modify the shape of the small bubble \((d = 3 \) mm\) and no preferential elongation in \( x \) or \( y \) direction is seen. The bubble is seen to rise along the center region of the container and it does not tilt as observed in Fig. 6.9 when no magnetic field is applied. At \( t^* = 5.75 \) the top of the bubble reaches \( z = 12.2d \) which is \( 0.4d \) below the location observed in the case with \( B = 0 \). The dashed lines in the figure are again approximate tangents to the top (outer surface) of the bubble, and the estimated average rise velocity with the graphical method is 2.4 which is slightly smaller than that seen without magnetic field.

Further increase of the magnetic field to 0.5 T makes the bubble shape more stable as the bubble remains at the center of the duct cross section (Fig. 6.12) at \( t^* = 7 \). No significant rotation of the bubble is observed.

The side views of the bubble reveal that the bubble is ellipsoidal and is slightly elongated in the magnetic
field direction \((x\) direction\). This is a result of the distribution of the Lorentz force which will be discussed in Section 6.4.2. The aspect ratio (the largest diameter along \(z\) divided by the largest diameter along \(x\)) of the bubble is calculated to be \(\sim 0.72\). The top of the bubble reaches \(z = 12.7d\) at \(t^* = 7\), whereas when there is no magnetic field the bubble arrives at the same position much earlier at \(t^* = \sim 5.75\).

**Dynamics of the 7 mm bubble**

Figure 6.7c showed a large oscillation in the rise velocity of the 7 mm bubble, implying large deformations of the bubble and oscillatory drag. However, bubble velocities in the initial stage are nearly the same with and without the magnetic field. This was also observed in a previous study [18]. Figure 6.13 shows the bubble shape with no magnetic field, in the time interval of \(t^* = 0\) to 1.8. Figure 6.13 shows that the bubble undergoes a significant deformation between \(t^* = 0\) to 1.25. The bubble first changes from a sphere to a “mushroom-head-like” shape at \(t^* = 0.75\) and then deforms into a squeezed (in \(z\) direction) ellipsoidal shape at \(t^* = 1.25\). During this time interval, the bottom half of the bubble moves up but the front of the bubble does not rise as much. This is due to the larger pressure on the top surface and the inability of the bubble to displace the liquid in front. It is important to note that here the rise velocity predicted by the graphical method (dashed line) will be smaller (if we draw the line at top) or larger (if we draw the line at bottom) than the integration method depending on where the line is drawn. The rise velocity curves with and without the magnetic field during the early stage \((t^* < 0.5)\) overlap because the bubble deforms slowly, and the speed of the surrounding liquid (although the internal gas velocity is quite large) and the generated magnetic resistance force in the surrounding conducting liquid are small. Thus, the effect of the magnetic field is small and the viscous and surface tension effects dominate during this early deformation.

The bubble shape after the initial deformation and the associated rise velocity of the 7 mm bubble with
$B = 0 \ T$ are shown in Fig. 6.14. It takes about 0.85 dimensionless time (equivalent to 0.023 second) for the

7 mm bubble to achieve 63\% of the terminal rising velocity (considering terminal velocity $w^* \approx 1.35$). The momentum response time $\tau_m$ of a 7 mm spherical bubble in Stokes flow can be obtained using

$$\tau_m = \frac{\rho g d^2}{18 \mu},$$

which gives $\tau_m = 2.4 \times 10^{-4}$ second. This response time is almost 100 times smaller than the observed response time of $2.3 \times 10^{-2}$ second. This result is also in agreement with a previous study\[18\] which observed that takes $0.02 \sim 0.03$ second for a large argon bubble ($E_o = 2.2$) rising in GaInSn to reach 63\% of the terminal rising velocity. The blue numbers in the sub-plots of Fig. 6.14 record the maximum number of cells in the cross section area of the bubble. The rise velocity is inversely related to cross-sectional area as indicated by those numbers.

**Shape oscillations of the 7 mm bubble**

For this bubble of larger diameter, the Eötvös number equals 2.80 larger than that of the 3 mm bubble ($E_o = 0.51$). Therefore more deformations are seen when compared with those of the 3 mm bubble. As shown in Fig. 6.14, the 7 mm bubble first undergoes a symmetrical deformation with the bottom part of the bubble moving up and forming an ellipsoid at $t^* = 2$. Subsequently, the bubble deforms asymmetrically with the largest cross section diameter of $1.7d$. Consequently, the frontal area of the bubble has increased,
exerting a larger drag on the bubble. The bubble rise velocity reaches a plateau at \( t^* = \sim 2 \). After \( t^* = 2 \), the bubble deforms differently and the frontal area reduces at \( t^* = 3.25 \), giving a higher rise velocity. At \( t^* = 5.0 \) the bubble is elongated along the diagonal of the duct with an increase in cross section. The bubble still remains at the center of the duct with a decrease in rise velocity. Further in time, at \( t^* = 8.24 \) the bubble cross sectional area is reduced again with an elongated axis in \( x \) direction. The elongation switches axes at \( t^* = 9.24 \), and the area of the cross section again increases.

As stated above, the oscillation of the rise velocity is linked to the bubble shape oscillation as well as the cross section area of the bubble. Fig. 6.14 shows that the bubble rise velocity varies with an approximate dimensionless cycle time period of \( t^* = 1.4 \). Using \( t = t^* g^{1/2} d^{1/2} \) we can convert this to a real time of 0.037 seconds, which corresponds to a frequency around 27 Hz. A previous experimental study\[44\] also found that when rising in purified water, the shape of a 5.5 mm air bubble oscillates with a frequency between 20 ∼ 30 Hz. Assuming the waves to be capillary waves traveling on the bubble surface, Lunde and Perkins\[45\] obtained an expression for the mode (2,0) shape oscillation frequency:

\[
f_{2,0} = \frac{1}{2\pi} \left( \frac{16\sqrt{2} \lambda^2}{(\lambda^2 + 1)^{3/2}} \right)^{1/2} \left( \frac{\gamma}{\rho_l (0.5d) ^2} \right)^{1/2}
\]

where \( \lambda \) is the ratio between the major and minor axes. In the present simulation without the magnetic field, \( \lambda = 2.1 \) is obtained from the side view of the 7 mm bubble. Substituting this in Eqn. 6.15 gives \( f_{2,0} = 28.3 \) which agrees very well with the present shape oscillation frequency of 27 Hz. However, \( \lambda \) varies a lot during the rise of 7 mm bubble, and therefore it is valuable to obtain the range of the \( f_{2,0} \). Figure 6.15 shows that \( \lambda \) increases between 1 to 3, \( f_{2,0} \) varies in between \( (25.6, 29.7) \). Therefore, in the argon-steel system, Eqn. 6.15

![Figure 6.15: Range of the shape oscillation frequency \( f_{2,0} \)](image)
is not very sensitive to the value of $\chi$, and the oscillation frequency is dominated by $\gamma^{1/2} \mu^{-1/2} (0.5d)^{-3/2}$ term. Without the magnetic field, a maximum $\chi = 2.63$ is found for the 7 mm bubble.

Figure 6.16 shows the side views of the 7 mm bubble rising without the magnetic field. From $t^* = 2$ to 3.25, the curvature at the front of the bubble becomes smaller while the curvature of the back becomes larger. The bubble thickness is slightly increased with time and the bubble remains in the center of the duct. The side views of the bubble from $t^* = 7.75$ to 9 indicate that the bubble deforms considerably, reflecting in

![Figure 6.16: Front views of the 7 mm bubble at different time $t^*$, without magnetic field](image)

the rise velocity history plot shown in Fig. 6.14. We can see that a larger thickness of the bubble is usually associated with a larger rise velocity. The graphical method is applied again to estimate the average rising velocity of the bubble during this time period. It is seen that the dashed line is basically tangent to the surface of the bubble between $t^* = 7.75$ to 8.25 but it passes inside of the bubble for $t^* = 8.50$ and 8.75. This indicates that a slightly smaller average rise velocity should be obtained during the time period from $t^* = 8.5$ to 9, which again agrees with the rise velocity curve shown in Fig. 6.14. These figures also show that the bubble is biased away from the centerline of the duct with the front of the bubble tilted and the bubble moving toward one side of the duct. The bubble also wobbles considerably during its rise. Although a larger deformation is seen for the 7 mm bubble, the bubble doesn’t deviate from the centerline too much and the front face of the bubble doesn’t tilt as much as that of the 3 mm bubble.

**Effect of magnetic field on shape of the 7 mm bubble**

As shown in Fig. 6.17, when a 0.2 T magnetic field is applied, the rise velocity of the bubble is initially reduced by a small amount and fluctuates around $W^* = 1.2$. As in the case without the magnetic field, first a symmetrical deformation of the bubble occurs and the bubble shape changes from a sphere to spheroidal disk shape (squeezed along $z$ direction) at $t^* = 2$ and a decrease in rise velocity is seen. At $t^* = 4$ the
bubble is seen to be elongated in the $y$ direction while remaining at the center of the duct. Then at $t^* = 8$ the bubble is seen to be elongated in the $x$ direction and the $x$ dimension of the bubble is $\sim 1.4d$, while the $y$ dimension is seen to be $1.2d$. At $t^* = 8.5$, the direction of elongation switches to the $y$ direction with the $y$-dimension becoming $1.5d$ and the $x$ dimension reducing to $1.1d$. The rise velocity reaches its local peak value of 1.35. The oscillation of the bubble shape continues with the major and minor axes switching directions, and the drag as well as rise velocity changing accordingly. The rise velocity at $t^* = 9.75$ is close to 1.39. The local wake interacts with the magnetic field, thus linking several nonlinear phenomena together.

Front views of the 7 mm bubble (in the $y-z$ plane) under the magnetic field of 0.2 T are shown in Fig. 6.18. These views show that the bubble rises mainly vertically and not much motion in the $y$ direction is observed. In the early stage ($t^* < 3.25$), the shape of the bubble is seen to be a squeezed ellipsoid but later a more complex and time dependent bubble shape is developed. This alternative elongation behavior seen in Fig. 6.17 can also be found through analysis of the side views in Fig. 6.18. Comparing the front views at $t^* = 7.75$ and $t^* = 9$, the thickness of the bubble did not change much but the dimension of the bubble along $y$ is larger at $t^* = 9$. Since the volume of the gas is conserved, the bubble is relatively longer in the $x$ direction at $t^* = 7.75$. These front views also show that the bubble is not tilted. Comparing the $z$ locations of the bubble we see that the bubble rise velocity is slightly smaller than the value with no magnetic field. The graphical method shows an average rise velocity of 1.2 during the time period between $t^* = 7.75$ and 9. This value is close to the average bubble rise velocity shown in Fig. 6.17.

Fig. 6.19 shows results for a further increase of the magnetic field to 0.5 T. We now observe a stable rise
of the bubble and no time-dependent oscillations in rise velocity are seen. The bubble is slightly elongated in the direction of the magnetic field (x direction). The bubble size in x direction is $1.24d$, and $1.16d$ in y direction. The shapes in the x-z and y-z planes are shown in the inset at $t^* = 12$ where it is seen that the bubble is well centered in the duct. The bubble oscillations seen earlier are suppressed and the steady rise velocity is reduced to 75% of that without a magnetic field. Comparing these side views with the side views of the 7 mm bubble under $B = 0$ and 0.2 T shown previously in Fig. 6.18, it can be observed that the bubble is thicker when the 0.5 T transverse magnetic field is applied.

The maximum number of cells across the bubble is 526 which is smaller than that when no magnetic field is applied (Fig. 6.14), however the rise velocity is 25% smaller than that without magnetic field. The reason for this is that the Lorentz force resists the motion of the liquid steel and makes it appear to be more viscous. Thus, more energy is required to move the surrounding liquid steel, causing a lower bubble rise velocity.

Figure 6.19: Bubble rise velocity and the associated top-view of the 7 mm bubble at different time (shown as blue square on the line plot) with $B = 0.5$ T.
6.4.2 Pressure and velocity fields and Lorentz force distribution

In the previous subsections, the bubble rise velocities and the deformations of the bubble were presented. Due to the close coupling between the velocity field and the induced Lorentz force, in this subsection we first show the pressure and velocity fields inside and adjacent to the bubble, and present the distribution of the Lorentz force, then demonstrate how the velocity field and bubble shape are affected by the Lorentz force.

The pressure field in the vicinity of the bubble is shown in Fig. 6.20 for the three cases of the 7 mm bubble. The pressures at the top and bottom of the rising bubble are shown in text. It can be seen that the maximum pressure difference increases with increasing $N$, in good agreement with a previous result.[18] This larger pressure drop across the bubble increases the drag acting on the bubble, and thus reduces its rising speed. Low pressure regions exist on the sides of the bubble, where maximum curvature is found. These pressures have an arbitrary level, hence only the difference is important.

The Lorentz force is generated due to the motion of the surrounding liquid (see Eqn. (6.2)). Since the bubble is insulated, no current can pass through the bubble and therefore the damping effect of the Lorentz force can only affect the bubble by modifying the velocity of the surrounding conducting liquid. The velocity field of the gas phase and the surrounding liquid phase under different magnetic field values for the 7 mm bubble are shown in Fig. 6.21 at $t^* = 2.5$. In all three cases, a recirculation pattern can be seen at the boundary of the bubble with streamlines pointing outwards at the top half of the bubble and pointing inwards at the bottom half of the bubble. Comparing the cases of $B = 0$ and $B = 0.2$ T, when the magnetic field is applied, the maximum vertical velocity $w^*$ at the bottom of the bubble is reduced from 1.6 to 1.3. With further increase of the magnetic field to 0.5 T, the bubble becomes thicker and less squeezed in vertical direction. The maximum value of $w^*$ of the liquid outside of the bubble is reduced from $-0.8$ to $-0.4$. The
velocities inside the bubble do not generate much Lorentz force because they are nearly insulated, therefore not much velocity reduction is seen inside of the bubble.

The contours of $z$ direction velocity ($w^*$) and path lines in the $y-z$ plane at $t^* = 8.75$ for different magnetic fields are presented in Fig. 6.22. The recirculation is still present for all three values of $B$, but the shapes of the bubble and the rise velocities behind the bubble are different. When no magnetic field is applied, the bubble is no longer symmetrical and rises as a wobbling disk with unsymmetrical vortices behind. The $w^*$ velocity behind the bubble is also unsymmetrical and it shows that $w^*$ velocity at the right side ($y > 3$) is higher than the left side. The bubble moves biased to the left side (towards the $y-$ direction).

Considering the upcoming liquid at the bottom of the bubble as an impinging flow on the bubble, the flow below to the right half of the bubble has a higher velocity. When the flow impinges on the bottom of the bubble, the bubble is pushed upward. Since the right side has a larger velocity, a larger pressure on the bubble is exerted and the bubble is biased to the left side. However, when the magnetic field is applied, there are no vortices shed and the flow velocity in the liquid is reduced with less asymmetry in the path of the bubble. Thus, the rise path is more rectilinear.
The contours of \( u^* \) velocity in the central \( x-z \) plane and \( v^* \) velocity in the central \( y-z \) plane for the three different magnetic field values at time \( t^* = 2.5 \) are presented in Fig. 6.23. Fig. 6.23(a) and (d) show that when no magnetic field is applied, the distribution of \( u^* \) velocity in the central \( x-z \) plane is similar to the distribution of \( v^* \) velocity in the central \( y-z \) plane, implying symmetry. When a magnetic field of \( B = 0.2 \) T is applied, Fig. 6.23(b) and (e) show still nearly similar distributions as those without the magnetic field. However, with further increase of \( B \) to 0.5 T, a different velocity distribution is seen in Fig. 6.23(c) and (f). Comparing the two figures, the high \( u^* \) velocity region in \( x-z \) plane is larger than the high \( v^* \) velocity region in the \( y-z \) plane, with larger maximum \( u^* \) than maximum \( v^* \). That the horizontal motion of the fluid in the \( x \) direction is more intensive than that in the \( y \) direction. In the momentum equations (Eqn. (6.2)), although the induced Lorentz force \( F_L \) consists of three components \( (F_{Lx}, F_{Ly}, F_{Lz}) \), the \( x \) component of the force \( F_{Lx} \) equals zero since it is parallel with the magnetic field direction. Therefore, the Lorentz force cannot directly affect the motion in the \( x \) direction. However, the damping effect of the Lorentz force in \( y \) and \( z \) directions affects the velocities in those directions and modifies the \( x \) direction velocity by continuity condition. The distribution of \( v^* \) velocity in the central \( y-z \) plane is different from the other cases with \( B = 0 \) or 0.2 T, and the contours of \( v^* \) show two tails following the bubble. Comparing Fig. 6.23(a), (b) and (c) we can see that the \( u^* \) velocity behind the bubble is significantly increased at \( B = 0.5 \) T and it is larger.
than the cases of $B = 0$ and 0.2 T. The Lorentz force does not directly affect the flow inside the bubble due to the almost zero conductivity of the gas, but under the strong magnetic field of 0.5 T Fig. 6.23(c) and (f) show that the circulation inside the bubble in the $y$-$z$ plane is reduced compared to the circulation in the $x$-$z$ plane. This is caused by the suppression of $y$ direction motion of the surrounding fluid. Therefore, the magnetic field can also reduce the circulation inside of the bubble indirectly by affecting the surrounding liquid.

Since the Lorentz force modifies the velocity distribution, in Fig. 6.24 we show the distributions of Lorentz force in the $y$-$z$ plane ($x/d = 3$) for the two different $B$ values at the same dimensionless time $t^* = 2.5$. The first plot in Fig. 6.24 showing contours of the $y$ component of the Lorentz force together with lines shows the direction of the Lorentz force. It is seen that on the top half of the bubble the force is pointing towards the inside of the bubble and tries to squeeze the bubble along $y$ and $z$ directions. However, at the bottom half of the bubble the $y$ component of the Lorentz force is positive on the right side but negative on the left side, which means that the force is trying to pull the liquid away from the bubble. The contour plots of $z$ component of the Lorentz force shows that the Lorentz force is pushing the liquid at top of the bubble downward and also pulling liquid downward at the bottom of the bubble. The force also decelerates the steel flow at the side of the bubble, thus diminish the recirculation. All these effects can be more clearly seen when $B$ is 0.5 T. The contours of $F_y^*$ also show that the force tends to push the liquid inward in the upper potion of the bubble and pull the liquid away from the bubble in the bottom potion. The two tails in the contours of $F_y^*$ are caused by the direction of the $y$ component of the velocity behind the bubble. The contour plot of $F_z^*$ at $B = 0.5$ T shows that the Lorentz force resists the rise of the bubble. In the $y$-$z$ plane the maximum $F_z^*$ at the bottom of the bubble is not at the center but slightly biased to one side.

Consistent with earlier studies [18, 24, 27–29] we have also seen that the bubble is elongated along the magnetic field direction. However, the mechanism by which the bubble is elongated has not been

![Figure 6.24: Contour plots of $F_y^*$ and $F_z^*$ at $t^* = 2.5$ and direction of Lorentz force, for bubble $d = 7$ mm at middle plane $x/d = 3$](image)
previously fully understood. Figure 6.25 shows the isosurfaces of force in y direction \( F_y^* \) and contours in two different planes. Analysis of these plots demonstrates that the bubble’s elongation should be rather expressed as “bubble compression” in the perpendicular plane, and the compression is caused by the anisotropic distribution of the Lorentz force. The left plot in Fig. 6.25 shows one quarter of the bubble (represented by

Figure 6.25: (a) Vectors of Lorentz force in y-z plane and x-z plane and eight isosurfaces of constant \( F_y^* \). (b) Contours of \( F_y^* \) at top and bottom half of the rising 7 mm bubble, \( B = 0.5 \) T and \( t^* = 8.75 \).

a blue surface at the center) and the isosurfaces of constant \( F_y^* \). In addition, the Lorentz force vectors in the quarter planes (y-z plane and x-z plane) are shown. The isosurface plot shows that \( F_y^* \) reaches its maximum value in the y-z plane and decays as we rotate it towards becoming the x-z plane. This is because the velocity component in y direction is decreasing, and in x-z plane the \( F_y^* \) is zero due to it being nearly symmetrical. Therefore only \( F_x^* \) component exists in x-z plane. Hence the force vectors in x-z plane only point upward or downward depending on the direction of the z velocity. However, due to the existence of \( F_y^* \) in the y-z plane the vectors are in y and z directions. As the bubble rises in the duct, the top of the bubble continually pushes away the surrounding liquid and if no transverse magnetic field is applied, an axisymmetric shape of the bubble is expected. In that case the liquid should be pushed away uniformly in the radial direction at the top half of the bubble and pulled liquid inward at bottom half of the bubble. Thus, there should be no difference between x and y directions of the bubble shape, unless instabilities trigger and lead to an asymmetrical shape. After applying the magnetic field in the x direction, a Lorentz force component \( F_y^* \) is
generated in the domain which acts as an additional resistance force along $y$ direction and tries to prevent the liquid being pushed away along $y$ direction. This leads to an additional compressive force that acts on the bubble. The bubble cannot push the surrounding liquid away along $y$ direction as easily as it can push liquid along the $x$ direction. Therefore the bubble ends up shorter in $y$ direction compared to $x$ (magnetic field) direction. It is therefore appropriate to interpret that the bubble “elongation in the magnetic field direction” to be actually caused by a compression in the direction perpendicular to the field.

6.4.3 Flow structures behind the bubble

The non-rectilinear motion of the bubble has been previously related to the structure of the wake formed behind the bubble [18, 43, 46]. It was proposed that these instabilities in the wake cause an asymmetrical flow behind the bubble and further lead to a zig-zag or helical motion of the bubble. In this section, we present the observed wake structures behind the bubble and discuss the changes caused by the magnetic field on these structures. Fig. 6.26 shows the front and side views of the vorticity magnitude of a value $|\omega^*| = 3$ at a representative time $t^* = 5.75$ for the three different magnetic field values. These plots show that the wake structure behind the bubble has a “tail-like” shape. A comparison of the lengths of the tails shows that with increase of the magnetic field strength, the wake becomes smaller. The reduction of the wake region is caused by two effects: (1) the external magnetic field induces a Lorentz force which increases the drag experienced by the bubble, causing a reduction in rise velocity and consequently slower motion of the
surrounding liquid and (2) the flow motion in the wake region is suppressed by the induced Lorentz force.

Isosurfaces of $\omega^*_z = \pm 1$ at $t^* = 5.75$ for the 3 mm bubble are shown in Fig. 6.27. These figures show that $\omega^*_z$ surrounding the bubble alternates in sign, which indicates that the rotation of the fluid in the $z$ direction consists of alternating rotating pairs. In the case without the magnetic field, the side views ($y$-$z$ plane) show that the bubble is slightly biased away from the center and rises towards the $y-$ side of the duct. The $z$ vorticity is seen to be unsymmetrical and the tail on the $y+$ side is slightly longer. The front view ($x$-$z$ plane) shows the bubble is biased from the center and rises towards the $x+$ side. Again, the $z$ vorticity is seen to have a longer tail on the opposite side ($x-$ side). After applying the magnetic field, the bias of $z$ vorticity disappears and the bubble is seen to rise rectilinearly. Surprisingly, upon further increase of the magnetic field to 0.5 T the affected region of the $z$ vorticity is seen to become larger, indicated by a longer tail following the bubble. Figure 6.27 also shows that with $B = 0.5$ T the $z$ vorticity behind the bubble does not alternate as many times as seen for $B = 0$ and 0.2 T. However, the top views of the $z$ vorticity in Fig. 6.28 show that in all these cases there are 4 rotation pairs in the front and surrounding the bubble.

The isosurfaces of $|\omega^*| = 4$ at $t^* = 9$ for $d = 7$ mm and different magnetic fields are shown in Fig. 6.29. With the larger bubble diameter, the wake structures behind the bubble are more complex and hairpin structures are seen. [14, 29, 47] When no magnetic field is present, the isosurface shows interconnected hairpin structures behind the bubble and they persist even after $5d$ behind the rising bubble. These structures indicate that a larger shear is experienced by the bubble. These hairpin structures are probably generated by the shape oscillations of the bubble, as no such structures are seen for the 3 mm bubble or in the initial
rise of the 7 mm bubble. The oscillatory bubble shape and complex wake structures behind the bubble lead to unsteady motion of the bubble. After a strong magnetic field is applied, the bubble shape is stable and the hairpin structures in the wake are suppressed. After applying a magnetic field of 0.2 T along \( x \) direction,

Figure 6.28: Top views of the bubble (green), isosurfaces of \( \omega_z^\ast = 1 \) (bright yellow) and \( \omega_z^\ast = -1 \) (dark blue) at \( t^\ast = 5.75 \) of 3 mm bubble for \( B = 0T, B = 0.2T \) and \( B = 0.5T \) (from left to right)

Figure 6.29: Front and side views of the bubble (green), isosurfaces of \( |\omega^\ast| = 4 \) (pink color) at \( t^\ast = 9 \) for 7 mm bubble with \( B = 0T, B = 0.2T \), and \( B = 0.5T \)

the vortex behind the bubble is compressed and the region affected in \( x \) direction is slightly larger than that along \( y \) direction. Although some hairpin-like structures are seen, they are smaller and more elongated in the magnetic field direction (\( x \) direction). The alternate shedding of these hairpin-like vorticities is related to the zigzag motion of rising bubbles. [47] With further increases of magnetic field strength to 0.5 T (\( N = 0.68 \)), the complex wake structures behind the bubble disappear due to the flow getting damped by the Lorentz force, and the bubble rises straight upward.

Figure 6.30 shows the isosurfaces of \( \omega_z^\ast = \pm 1 \) at \( t^\ast = 9 \) for the 7 mm bubble. A pattern of alternating \( \omega_z^\ast \) is seen again but the structure behind is more complex than that seen behind the 3 mm bubble. With a magnetic field of 0.2 T, it is seen that the isosurfaces are elongated in the magnetic field direction (\( x \)
direction). This effect is caused by the fact that by applying the magnetic field along $x$ direction the $y$ and $z$ velocities in the surrounding fluid are reduced, thus damping the flow perpendicular to the magnetic field direction. The damping effect of the external magnetic field of different strengths has been earlier studied in detail for flow in a driven cavity. [38] With increase of the magnetic field strength to 0.5 T, the complex wake structure is seen to nearly disappear. It is seen that the front view of the isosurface is wider (along $x$ direction) in the region close to the bubble, while in the downward region the isosurfaces are bundled. However, the wake along $y$ direction is not compressed but slightly spreads out.

### 6.4.4 Modified drag coefficient

In the steel casting industry, argon bubbles are often injected during the continuous casting process to prevent clogging and to remove inclusions (i.e. aluminum oxide). [3, 48] A transverse static magnetic field up to 0.3 T is often used to optimize the flow pattern in the mold region. [49, 50] Several numerical simulations [48, 50–52] have been previously carried out to understand the transport of these argon bubbles in the caster. However, most models do not include the effect of Lorentz force on the drag coefficient. Since the present work shows that ignoring this effect may cause errors in the bubble velocities, a modified drag coefficient is calculated using present simulations.

When the steady state is reached, the drag force acting on the bubble is balanced with the buoyancy force. For a fixed bubble volume, the buoyancy force is independent of $N$. Therefore the total drag force
exerted on the bubble is a constant. Hence, one can write

\[ C_D w^2 = C_D w_0^2 = \text{Constant}, \] (6.16)

where \( C_D \) is the drag coefficient and \( w \) is the bubble rise velocity, subscript “0” denotes the value obtained when no magnetic field is applied (\( N = 0 \)). The rise velocities of the bubble are known from the simulations, and then Eqn. (6.16) is used to compute the estimated new \( C_D/C_{D0} \). The results are presented in Table 6.7.

Comparing the results for 5 and 7 mm bubbles with the same Stuart number, the predicted \( C_D/C_{D0} \) is slightly different, especially when \( N \) is small. This difference may caused by the bubble shape and its oscillation. With \( N = 0.085 \), the 5 mm bubble rises with out shape oscillation, but the shape of the 7 mm bubble still oscillates and leads to a oscillatory rise velocity. With \( N = 0.649 \), both 5 and 7 mm bubbles rise rectilinearly and the calculated \( C_D/C_{D0} \) for both cases are very close.

Previous experimental work [53] of conducting fluid flow past rigid spheres suggests that

\[ C_D = C_{D0} \left( 1 + 0.7\sqrt{N} \right), \] (6.17)

is suitable for the range \( 17.6 < Re < 332 \) and \( N < 2.5 \times 10^3 \). Present study shows the shape of the larger argon bubble is not spherical, and the \( Re \) number is higher than 1000, therefore Eqn. (6.17) cannot be used directly. To obtain a modified drag coefficient, a curve fitting of the values in Table 6.7 yields:

\[ C_D = \begin{cases} 
C_{D0} \left( 1.0 + 1.50N + 7.06N^2 \right), & \text{if } 0 \leq N < 0.245; \\
1.8C_{D0}, & \text{if } 0.245 \leq N < 0.65.
\end{cases} \] (6.18)

The above curves suggest that \( C_D/C_{D0} \) increases with \( N \) when \( N < 0.25 \), and it reaches a plateau of \( C_D/C_{D0} = 1.8 \) for \( N > 0.25 \). The above relation includes the effect of bubble shape deformation and works for low Morton number (\( Mo = 1.3 \times 10^{-12} \)) system and \( Eo < 2.80 \). Equations (6.17) and (6.18) are also shown in Fig. 6.31 together with the simulation results. However, Eqn. (6.17) under-predicts the modified drag coefficient when the magnetic effect is strong. Possible reasons for this discrepancy are: (1) bubble is
deformable and the shape is not spherical; (2) the bubble $Re$ is higher than the upper bound of 332. It is important to note that the present study shows applying a transverse magnetic field increases the drag coefficient by as much as 1.8 times.

### 6.5 Summary

In this paper we have studied the three dimensional dynamics of an argon bubble rising in molten steel in the presence of a transverse magnetic field. A VOF interface tracking method with sharp surface force treatment is used to reduce spurious velocities near the bubble. The code was validated by simulating the rise of an air bubble in water and, the rise of an argon bubble in GaInSn. The predicted rise velocities matched very well with other published results. Then six simulations of argon bubbles of two different sizes rising in molten steel were performed with the shapes of the bubble well resolved. The results show that an external magnetic field reduces the rise velocity, suppresses bubble shape oscillations and leads to a more rectilinear path. For a 3 mm argon bubble rising under a magnetic field of 0.2 T, the terminal rise velocity is seen to slightly increase compared with that of the no magnetic field case. This is caused by the fact that the rising path of the bubble is more rectilinear. Further increase of the magnetic field reduces the rise velocity. For a 7 mm bubble, the results show that the bubble undergoes large deformations and shape oscillations. The oscillatory bubble shape further affects its rise velocity as well as the wake behind. During the oscillations, the bubble rises slower with a larger cross section area. Shape oscillations also generate hairpin structures in the wake. Without the magnetic field, the shape of a 7 mm bubble oscillates at a frequency around 28 Hz. The simulations also show that with a large magnetic field strength, the unsteady shape deformations are suppressed and the hairpin type complex wake structures behind the bubble almost disappear. The results also show that although the gas is not conducting, by modifying the fluid velocity adjacent to the bubble, the applied magnetic field reduces the circulation inside the bubble. The bubble is seen to be compressed in the direction perpendicular to the applied magnetic field direction. The observed results are a direct result
of the Lorentz force distribution which increases the resistance to flow motion in the plane perpendicular to the magnetic field. A correlation of $C_D/C_{D0}$ with the Stuart number is presented.

### 6.6 References


Chapter 7

Effect of EMBr on Ar Bubble Capture using LES Model

In this chapter, a two-way coupled Eulerian-Lagrangian computational model was applied to simulate turbulent flow of the molten steel and the transport and capture of argon gas bubbles into the solidifying shell in a continuous slab caster. The computational domain was discretized with more than sixteen million cells, and the turbulent flow was modelled using Large Eddy Simulation (LES) model. The trajectories of 1.2 million argon bubbles were tracked by solving the transport equations for each individual bubble. A previously validated advanced capture criterion was used to predict the capture of argon bubbles. All the equations are solved in parallel on Graphic Processing Units (GPUs) using an in-house code CUFLOW. Less bubbles were captured in this LES model when compare with previous RANS model [1, 2]. The predicted captured bubble size agrees well with plant measurement.

7.1 Introduction

In steel continuous casting, argon gas is usually injected to prevent clogging [3–5]. However, argon bubbles also affect the flow pattern, and may become entrapped to form defects, such as blisters and slivers, in the rolled steel product [6, 7]. After entering the Submerged Entry Nozzle (SEN), argon bubbles are carried by the turbulent flow through the SEN and into the mold cavity region, where they greatly affect the flow pattern, surface level fluctuations, and slag entrainment. Bubbles entering the mold region end up at three locations: (1) most large bubbles reach the top surface, pass through the slag layer and escape harmlessly to the atmosphere [1, 2]; (2) some are captured near the meniscus and lead to surface defects; (3) some small and tiny bubbles are captured deep in the caster and cause internal defects. During the transport, moving bubbles collect non-wetting inclusion particles, such as alumina. If such a bubble is captured into the product, the layer of inclusions covering its surface will lead to large oxide clusters, which cause severe sliver defects [8, 9].

Computational models and water models have been used to study the two-phase flow of argon and molten steel in the SEN and mold region of the continuous caster [1, 2, 8–25]. Both Eulerian-Eulerian [11, 19–21, 26]
and Eulerian-Lagrangian [10–16, 18, 27] approaches have been used in computational models. With high gas fractions, three-dimensional and two-way coupled models are more appropriate as the argon gas affects the steel flow and vice versa. [10–12, 22] Increasing Ar gas causes increased upward flow near the SEN and tends to reverse the classic double-roll flow pattern to single-roll with surface flows away from the SEN towards the narrow face. [11, 20, 21]. Asymmetric, oscillating flow is observed if gas fractions are excessive [3, 28]. Argon bubbles in steel are reported to be larger than air bubbles in water [22, 29], and increasing gas flow generates more and larger bubbles. [3, 28, 29] Recently, MUSIG model [30] has been used to model this multi-phase flow problem and it allows Eulerian-Eulerian approach to include the effect of local bubble size. [22, 31]

Several studies [12, 14–18, 23, 25, 32–34] have investigated the capture fraction and distribution of inclusion particles in continuous casting, but none of these studies of particle transport included the coupled effects of Ar gas on the flow field. Recent studies [1, 2] used Reynolds-Averaged Navier Stokes (RANS) model with Lagrangian approach to investigate the transport and capture of argon bubbles. The chaotic motion of individual particles in turbulent flow dispersion behavior was mimicked using a widely accepted Random Walk Method. [12, 13, 16–18, 22–25, 27, 32, 33] This approach includes the effect of bubbles with different diameters on the local fluid flow. By tracking the motion of each individual bubble and using an advanced capture criterion [15, 16, 24], they found ~85% of small (< 0.08 mm) bubbles are captured. A very small fraction of large bubbles is captured (< 0.02%). The predicted capture location and size of bubbles matched with plant experiments in most places except region very close to the meniscus, where the capture rate is slightly under predicted by the model. This may due to absence of the mechanism of bubbles captured by hooks. [7]

The velocity across the top surface of the mold is an important parameter affecting defect formation. A very small velocity causes reduced heat transfer and leads to hook formation, meniscus freezing, and other surface defects. On the other hand, if the top surface velocity is too large, the resulting turbulence and shear layer instability may entrain slag and form inclusions in the final product. Electromagnetic braking (EMBr) system is often used to dynamically control the mold flow. In this approach, a DC current is applied to a coil, a static magnetic field is generated which in turn induces a Lorentz force field that acts against the flow. Based on the DC electromagnets shape and location, there are usually three types of static magnetic field configurations: local [13, 35–39], single ruler [39–44] and double ruler [20, 44–48]. The difference between these configurations as well as the use of AC in the electromagnets are discussed elsewhere [49]. The double ruler configuration which is widely used in industry and commonly known as the Flow Control Mold (FC-Mold) (ABB Automation Technologies) [50]. In the double ruler configuration, two rectangular magnetic fields across the entire mold width are generated, with one positioned near the meniscus and the other below...
the nozzle ports[20, 45–49]. This configuration is able to slow down[45, 48] or to speed up[44, 47] surface velocities in the mold region, and has been reported to decrease high-frequency turbulent fluctuations[44] and to improve meniscus stability[47, 48].

Many researchers investigated the effect of double ruler EMBr on the fluid flow[20, 44–48], but only limited work studied its effect on argon bubbles or inclusion transport and capture.[13, 30, 51] With RANS approach and Lagrangian particle with random walk model, studies showed EMBr has little effect on the removal fraction of small bubbles. [13, 20] The random walk model assumes isotropic turbulence. At region close to the wall, this model gives same fluctuation velocities in all direction which may over predict the number of captured inclusions. Furthermore, both studies ignored the effect of conducting shell, but LES results showed that the conducting wall can help stabilize the flow and suppress the unsteady low frequency oscillation behavior of the flow.[43, 44] Neither of these studies considered the effect of magnetic field on the drag coefficient of the inclusions or bubbles. Previous VOF simulation of argon bubble rise in liquid steel under magnetic field [our bubble rise paper] showed the bubble shape can be affected by the field strength and the velocity can be reduced by 25% comparing with that of no magnetic field.

This paper investigates the transport and capture of argon bubbles using a LES model for turbulent flow and Lagrangian description for argon bubbles. The solidified shell is included in the computational domain, the effect of EMBr on bubble drag force is modeled through using a modified drag coefficient (see Chapter 6). The results are compared with previous RANS simulation and experiment.[1, 2]

7.2 Computational Models and Numerical Methods

A three-dimensional finite-volume computational model together with two-way coupled Lagrangian tracking method was applied to study the fully-coupled turbulent flow behavior and the transport phenomena of argon bubbles in a commercial continuous steel caster. The computational domain includes the slide gate, SEN and mold region (from meniscus surface to 2.65 m below meniscus), as shown in Figure 7.1 In the plant operation, 1300 mm wide slabs were casted at 1.5 m/min, the slide gate was 70% open[1, 2], as shown in Figure 7.2. The computational domain was discretized using ∼16 million hexahedral finite volume cells. Casting conditions are taken as described in the references[1, 2] and listed in Table 7.1.

The assumed shell thickness $s$ (mm) at any point $z$ (m) below the meniscus is plotted in Figure 7.3 and it is calculated as

$$s = k \sqrt{z/V_c}$$

where $V_c = 0.025$ m/s (1.5 m/min) denotes the casting speed and the constant $k = 3$ mm/s$^{1/2}$ is chosen to
Figure 7.1: Computational domain

Figure 7.2: Slide gate position, 70% open at $V_c = 1.5$ m/min
Table 7.1: Process parameters

<table>
<thead>
<tr>
<th>Process parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mold thickness ((L_t))</td>
<td>230 mm</td>
</tr>
<tr>
<td>Mold width ((L_w))</td>
<td>1300 mm</td>
</tr>
<tr>
<td>Slide gate opening area fraction</td>
<td>70 pct</td>
</tr>
<tr>
<td>SEN Submergence depth</td>
<td>160</td>
</tr>
<tr>
<td>Nozzle port downward angle</td>
<td>15°</td>
</tr>
<tr>
<td>Nozzle port area (width (\times) height)</td>
<td>65×83 mm²</td>
</tr>
<tr>
<td>Casting speed ((V_c))</td>
<td>1.5 m/min</td>
</tr>
<tr>
<td>Steel density ((\rho))</td>
<td>7000 kg/m³</td>
</tr>
<tr>
<td>Argon density ((\rho_p))</td>
<td>0.5 kg/m³</td>
</tr>
<tr>
<td>Steel dynamic viscosity ((\mu))</td>
<td>0.0063 kg/(m·s)</td>
</tr>
<tr>
<td>Argon dynamic viscosity ((\mu_p))</td>
<td>0.0000212 kg/(m·s)</td>
</tr>
</tbody>
</table>

match a break-out shell profile from the caster at Baosteel.

![Shell Profile](image.png)

Figure 7.3: Shell profile

### 7.2.1 Governing Equations for the Fluid Flow

In this work the turbulent flow in the SEN and mold is simulated by the LES model. The three-dimensional time-dependent Navier Stokes equations given below were solved:

\[
\nabla \cdot (\rho u) = \dot{s}
\]

\[
\rho \frac{\partial u}{\partial t} + \rho \nabla \cdot (u \otimes u) = -\nabla p + \nabla \cdot \left[ (\mu + \mu_{\text{sgs}}) \left( \nabla u + \nabla u^T \right) \right] + S_L + S_{\text{sink}} + S_p
\]

where \(\rho\) is the density of molten steel, and \(S_{\text{sink}}\) are the mass and momentum sink terms\([2, 15, 52]\) added to include the effect of the solidifying shell, \(u\) is the velocity vector, \(p\) is a modified static pressure which includes the normal stresses, \(S_p\) is the source terms due to two-way coupled Lagrangian particle tracking, \(\mu\)
is the dynamic viscosity of the molten steel and \( sgs \) is the eddy viscosity that represents the subgrid stress. In this study, \( sgs \) is modeled by the coherent-structure Smagorinsky model (CSM) sub-grid scale (SGS) model\[53\]. In the CSM SGS model, \( sgs \) is computed as:

\[
\mu_{sgs} = \rho \nu_{sgs} = \rho (C_s \Delta)^2 \sqrt{2 ||S||}
\]

(7.4)

where \( \Delta \) is the size of the cubic cell and \( S \) is the rate-of-strain tensor given by: \( S = 1/2 (\nabla u + \nabla u^T) \). \( C_s^2 \) is calculated locally by the following equations:

\[
C_s^2 = C_{csm} |Q/E|^{3/2} (1 - Q/E)
\]

(7.5)

\[
Q = 1/2 \left( ||W||^2 - ||S||^2 \right)
\]

(7.6)

\[
E = 1/2 \left( ||W||^2 + ||S||^2 \right)
\]

(7.7)

where \( C_{csm} = 1/22 \) is a model constant and \( W = 1/2 (\nabla u - \nabla u^T) \) is the vorticity tensor. The CSM model appropriately damps the eddy viscosity in wall boundary layer regions and also automatically incorporates the effect of anisotropy induced by the applied magnetic fields on the subgrid scales\[54\]. Therefore, no additional modifications to account for anisotropic subgrid effects are needed or added. This model has been successfully tested and used previously in predicting fluid flow in steel casters with magnetic fields\[43, 47\].

### 7.2.2 Lagrangian Model for Argon Bubbles

In plant operation, Argon gas was injected through porous refractory in the upper tundish nozzle and SEN to prevent clogging. In this work, the Ar gas volume was distributed into bubbles according to a Rosin-Rammler\[55\] size distribution with average bubble diameter \( d_{mean} = 3 \) mm and spread parameter \( \eta \) was taken as 4. These are the same parameter used in previous RANS simulations\[1, 2\]. These two parameters are chosen based on calculations with a previously-validated two-stage model of Ar injection into downward flowing steel\[3, 29\] and adjusted to account for increased surface tension in steel/argon using a relation from previous measurements\[3, 29\] and recent measurements\[56\] of bubble distributions. In the simulation, 407 bubbles are randomly placed slightly below the slide gate in every 0.01 second. The number distribution of bubbles are shown in Figure 7.4. For a 30 seconds LES simulation, the trajectory of 1.2 million bubbles are
Figure 7.4: Number of bubbles injected in every 0.01 second

tracked by solving the particle motion equations for each individual particle:

$$\frac{dx_p}{dt} = u_p$$  \hspace{1cm} (7.8)

$$m_p \frac{du_p}{dt} = F_{pT} = \Sigma (F_{pD} + F_{pL} + F_{pP} + F_{pA} + F_{pB})$$  \hspace{1cm} (7.9)

where $u_p$ and $x_p$ are the particle location and velocity, respectively. The total force, $F_{pT}$, acting on an individual bubble is the summation of five forces: drag force $F_{pD}$, lift force $F_{pL}$, added mass force $F_{pA}$, pressure gradient force $F_{pP}$ and buoyancy/gravity force $F_{pB}$. Note that the motion of each bubble is not a function of other bubbles, thus this data independence property is an ideal candidate for parallel processing.

The response time of particle or droplet in fluid is:

$$\tau_p = \frac{\rho_p d_p^2}{18 \mu_f}$$  \hspace{1cm} (7.10)

A smaller response time means the dispersed phase need a very short time to respond to a change in velocity. The response time for 1 mm bubble in liquid steel is $\sim 10^{-6}$. Thus, argon bubbles response to the motion of steel flow very quickly.

**Forces on Argon Bubbles**

The drag force acting on a single bubble is calculated as:

$$F_{pD} = \frac{C_D}{24} \frac{18 \mu}{d_p^2} V_p (u_{fp} - u_p) Re_p$$  \hspace{1cm} (7.11)
where $V_p$ is particle volume, $Re_p$ is particle Reynolds number, $u_{Fp}$ is the fluid velocity at particle location, and $C_D$ is the drag coefficient. Different from solid particles, the shape of argon bubbles deform during transport in the turbulent flow. The shape deformation of bubbles causes variation in the drag force. In this study, the drag coefficient proposed by Kuo et al.\cite{57} are used, and the drag coefficient are computed based on bubble Reynolds number and Weber number $We$ as follows:

\[
C_D = \begin{cases} 
16Re_p^{-1} & \text{if } Re_p \leq 0.49 \\
20.68Re_p^{-0.643} & \text{if } 0.49 \leq Re_p \leq 100 \\
6.3Re_p^{-0.385} & \text{if } 100 \leq Re_p \\
We/3 & \text{if } 100 \leq Re_p \text{ and } We < 2065.1We^{-2.6} \\
8/3 & \text{if } 100 \leq Re_p \text{ and } We < 8
\end{cases}
\quad (7.12)
\]

where the Weber number $We = \rho_l d_p \gamma^{-1} |u - u_p|^2$ and $\gamma_{lg} = 1.2 \text{ N/m}$ is the surface tension between argon and steel. This model take into account the bubbles shape. Figure 7.5a compares the predicted terminal rise velocities of a single air bubble rise in water and argon bubble rise in liquid steel by using this model, with measurements \cite{58} of air bubbles rise velocity in distilled and contaminated water. Figure 7.5b shows the results on a Grace diagram. \cite{59, 60} The Eötvös number $Eo = (\rho_l - \rho_g) gd^2 \gamma^{-1}$ (when $\rho_l \gg \rho_g$, the Eötvös number can be approximated by the Bond number $Bo = \rho_l gd^2 \gamma^{-1}$, where $\gamma$ denotes the surface tension) reflects the importance of surface tension force to gravitational force. It is used together with Morton number $Mo = g\mu/4(\rho_l - \rho_g) \gamma^{-3}$ to characterize the shape of bubbles. For a given pair of liquid and gas, $Mo$ is fixed and $Bo$ depends on the bubble size. The constant Morten number lines and bubble shapes are obtained from previous studies. \cite{59, 60}

In a conducting liquid, bubbles move slower when an external magnetic field is applied. Using a very fine mesh and VOF method, Jin et al. \cite{61} (see Chapter 6) studied the dynamics of single argon bubble rise in steel with horizontal magnetic field applied. A modified drag coefficient $C_{D-MHD}$ is extracted from the results:

\[
C_D = \begin{cases} 
C_{D0} \left(1.0 + 1.50N + 7.06N^2\right), & \text{if } 0 \leq N < 0.245; \\
1.8C_{D0}, & \text{if } 0.245 \leq N < 0.65.
\end{cases}
\quad (7.13)
\]

Where $N = Ha^3/Re_p$ is the Stuart number, and $Ha$ is the Hartmann number:

\[
Ha = Bd_p \sqrt{\sigma/\mu}
\quad (7.14)
\]

This modified drag coefficient is used to compute the drag force when EMBr is turned on. More details
Figure 7.5: (a) Predicted rise velocities of air bubble in water and argon bubble in steel (b) Predicted bubble rise velocity on Grace Diagram.

about modified drag coefficient can be found in Chapter 6 and reference.

The lift force, pressure gradient force, added mass force and buoyancy force are computed as following:

\[ F_{pL} = C_L \rho V_p (u_p - u_{Fp}) \times (\nabla \times u_{Fp}) \]  
(7.15)

\[ F_{pP} = \rho V_p \frac{Du_{Fp}}{Dt} \]  
(7.16)

\[ F_{pA} = 0.5C_V \rho V_p \left( \frac{Du_{Fp}}{Dt} - \frac{du_p}{dt} \right) \]  
(7.17)

\[ F_{pB} = gV_p (\rho_p - \rho) \]  
(7.18)

where \( C_L \) and \( C_V \) are the lift and added mass coefficient, respectively. In this study, \( C_V \) is taking as 1.0[62], and \( C_L \) is calculated using the Legendre and Magnaudet Lift force model[63]:

\[ F_{pL} = C_L \rho V_p (u_p - u) \times (\nabla \times u) \]  
(7.19)

\[ C_L = \sqrt{\left( C_{L_{lowRe}} \right)^2 + \left( C_{L_{highRe}} \right)^2} \]  
(7.20)
where, $C_{L}^{lowRe}$ and $C_{L}^{highRe}$ are computed following Eqn. (2.23)

$$C_{L}^{highRe} = \left( \frac{1}{2} \right) \frac{1 + 16Re_p^{-1}}{1 + 29Re_p^{-1}}$$

and

$$C_{L}^{lowRe} = 6\pi^{-2} (SrRe_p)^{-0.5} J'(\varepsilon)$$

(7.21)

$$J'(\varepsilon) = J(\infty) (1 + 0.2\varepsilon^{-2})^{-3/2} \quad \text{and} \quad \varepsilon = \sqrt{SrRe_p^{-1}}$$

(7.22)

where $J(\infty) = 2.55$ is a constant and $Sr$ is the shear rate of the fluid at the particle location.

In this model, the velocity of fluid at particle locations, $u_{Fp}$, are required to compute the forces acting on the particle. However, the center of particles and the center of finite volume cells are not overlapped in most cases. Thus, the fluid velocity at each particle location is computed using a second degree Lagrange polynomials interpolation method[64, 65], which involves 27 surrounding fluid grid points.

**Effects of SGS on Argon Bubble Dispersion**

The dispersion of bubbles from the SGS motion are include by adding a SGS velocity to the fluid velocity at the particle location. In the approach, the SGS kinetic energy $k_{sgs}$ is computed in each finite volume cell using the Yoshizawa SGS model[66]:

$$\nu_{sgs} = 0.066\Delta t_{sgs}^{1/2}$$

(7.23)

When evaluating the forces exerting on particles using equation (7.15) - (7.18), the resolved fluid velocity at particle location $\bar{u}_{Fp}$, is first obtained from Lagrange polynomials interpolation. Then a random fluctuation velocity $u_{sgs}$ is added to $\bar{u}_{Fp}$ to compute $u_{Fp}$:

$$u_{Fp} = \bar{u}_{Fp} + u_{sgs}$$

(7.24)

and

$$u_{sgs} = \zeta \sqrt{\frac{k_{sgs}}{3}}$$

(7.25)

where $\zeta$ is a random vector. Its components obey standard normal distribution with mean of 0 and standard deviation of 1.0. In each timestep, $\zeta$ is generated for each fluid cell and particles reside in the same cell use the same fluctuation velocity to compute $u_{Fp}$. In this study, $5.7 \times 10^{13}$ random numbers are generated for a 30 seconds LES simulation with timestep size of $2.5 \times 10^{-4}$ second. During the calculation, these numbers are generated in parallel using an Nvidia K40 GPU through the XORWOW[67] scheme which has a period larger than $6 \times 10^{57}$. 

173
Two-way Coupled Eulerian-Lagrangian Model

The motion of argon bubbles can affect the fluid flow through the source term ($S_p$) in the momentum equation. When there are multiple argon bubbles stay in a single cell, the source term is the summation of the contribution of each bubble/particle in that cell:

$$S_p = \sum_{i=1}^{n_i} S_{p,i} = -V_{cell}^{-1} \sum_{i=1}^{n_i} (F_{pD} + F_{pL} + F_{pA} + F_{pP})$$ (7.26)

where $V_{cell}$ is the cell volume and $n_i$ is the number of the particles contained in the finite volume cell.

Capture Criterion

To predict the capture location of the argon bubbles, an advanced capture criterion that based on force balance are implemented. This criterion is briefly discussed below, and the procedure of this advanced capture criterion is available elsewhere.[15, 16, 24] For small bubbles less than the Primary Dendrite Arm Spacing (PDAS), the particle can enter between the dendrite arms and be captured by entrapment. For bubbles/particles greater than the PDAS, the advanced criterion considers eight forces acting on a spherical bubble/particle touching three dendrite arms. This capture criterion has been validated by using a RANS and Lagrangian particle model to predict the number and capture locations of argon bubbles in a commercial caster and compare the results with plant measurements.[1, 2] Argon bubbles touches SEN walls will be bounced back with a negative velocity component and assume the perpendicular velocity is reduced to 50% as illustrated in Figure 7.6.

Figure 7.6: An argon bubble bounces back after hit SEN wall
7.2.3 Equations for Lorentz Forces

An electric potential method is used to compute the Lorentz force. This method exploits the fact that the induced magnetic field is much smaller than the externally-imposed magnetic field and therefore can be ignored.\cite{20, 37, 38, 40–45, 47, 68} The Lorentz force per unit volume is calculated by first solving a Poisson equation for the electric potential $\Phi$, then compute the current density $J$:

$$\nabla \cdot (\sigma \nabla \Phi) = \nabla \cdot [\sigma (u \times B)] \quad (7.27)$$

$$J = \sigma (-\nabla \Phi + u \times B) \quad (7.28)$$

$$S_L = J \times B \quad (7.29)$$

where $\sigma$ is the electrical conductivity. Note that both the molten steel and the solidified shell are conducting materials, so the above MHD equations must be solved in both the liquid steel and shell regions. Replacing the shell with an electrically-insulated wall boundary condition leads to incorrect results.\cite{43, 44} Thus, in this work, the above MHD equations are solved in the entire domain including the solid shell region. An insulated wall is applied at the exterior of the shell, considering that the slag in the mold / shell gap has very low conductivity.

7.2.4 The Solver

The above governing equations are solved with an in-house code CUFLOW \cite{69–72} which utilizes Graphics Processing Units (GPU). Current version of the code was written using CUDA Fortran, and has been extended to be able to work on multiple GPUs in parallel through the Message Passing Interface (MPI). \cite{72} In CUFLOW, fractional step method is used to solve the continuity and momentum equations. Pressure and electric potential Poisson equations are solved efficiently by a V-cycle multigrid method, and red-black Successive over Relaxation (SOR) on GPUs with over-relaxation parameter of 1.6. Details of the solution algorithm are available in reference \cite{70–72}. The particle transport equations are parallelized onto GPUs and solved with high efficiency. Figure 7.7 shows the solution procedure. CUFLOW has been validated and used to study the effect of magnetic field in a lid-driven cavity\cite{73}, argon bubble rise in liquid steel under magnetic field, and the flow in the mold region of continuous casters of steel with and without EMBr\cite{44, 47}.

175
7.3 Results and Discussion

7.3.1 Motion of liquid steel

The flow pattern of molten steel in the mold region critical to the final quality of product. Figure 7.8 shows the time-averaged flow pattern in the symmetry plane of SEN, center plane of the mold and 1 cm below the top surface. The liquid steel first enter the inner radius side of the SEN with a maximum speed of \( \sim 3 \) m/s, forms a recirculation region with length \( \sim 0.2 \) m on the outer radius side beneath the middle sliding plate. Two equal-sized swirls are seen at SEN bottom. The stream traces in the center plane \((y = 0)\) show a typical double role flow pattern. Near the top surface, the fluid moves almost parallel to the wide face with a speed of 0.25-0.3 m/s without any cross flow. This flow pattern predicted using LES is different from that predicted using RANS approach.[2] In previous RANS simulation, the top region of each half mold contains two big eddies: one beside the SEN and the other close to the narrow face. The eddy beside the SEN was driven by the buoyancy of argon gas. The eddy close to the narrow face was caused by nozzle jets impinge on the narrow face and split, sending some recirculating flow upwards and across the top surface toward the SEN.[2] The eddy beside SEN disappears in the transient LES simulation, owing to the buoyancy of rising argon gas is not strong enough to form a big eddy. In previous RANS approach, most argon bubbles traveled following a time-averaged flow pattern which causes concentration of momentum sources to be added into fewer cells that are near the SEN. However, the LES model captures the time dependent behavior of the swirls at the bottom of SEN. In LES, the swirls inside the port frequently switches directions, and the unsteady swirling
jet send argon bubbles to different locations at different time. However, the swirls inside the port predicted by previous RANS model was from inner to outer radius, which result in more bubbles exit the port near the inner radius. The flotation of those bubbles forms the eddy close to the SEN.

![Figure 7.8: Time-averaged flow field (a) inside SEN, (b) at the centerplane of mold and (c) 1 cm below top surface, without EMBr](image)

7.3.2 Transport of argon bubbles

Contours of steel flow and bubbles (within 2 mm in/out of the symmetry plane) in the symmetry plan of SEN \((x = 0)\) at time \(t = 24\) s are shown in Figure 7.9. Bubbles are randomly injected in a cylinder shape region (a rectangular in the 2D slice) below the slide gate with zero initial velocity. Most bubbles are carried downward by the high velocity steel flow. In the recirculation region below the slide gate, many bubbles move up into the slide gate and accumulated below the middle sliding plate which implies the form of gas pocket, which matches with water experiment. Since most region in the mold has a low argon gas volume fraction and this study is not focused on the high argon volume fraction region beneath the slide gate, collision or coalescence model is not included, therefore this model cannot directly predict shape of gas pocket. To study the formation of those argon bubbles, some other Eulerian approach is needed to resolve the interface of the gas and liquid steel which is beyond the scope of the presented work. No bubbles seen in the upper slide gate due to high speed downward moving steel flow flush bubbles downward.

Swirls are formed at the bottom of the SEN with different rotation direction, and these swirls affect the location at which argon bubbles exiting the port[1, 2], and several previous works have discussed the
relation of the swirl flow in the SEN with the bulk flow in the mold.[74, 74] Previous study [2] with RANS approach showed the swirl has downward velocity at inner radius region and most bubbles entered the mold close to the inner radius which led to biased capture of argon bubbles on wide faces. However, this study shows the swirls are unsteady and frequently change their rotation direction, and sometime two equal sized swirls are seen. Figure 7.10 (a) and (b) shows contours of $y$-velocity in the symmetry plane at SEN bottom at $t = 23.5$ and 24 s. At $t = 23.5$ s, the white spot above the bottom well of SEN shows flow moves from outer radius toward inner radius. While at $t = 24$ s, no in/out of plane motion is seen and two equal sized swirls formed at the bottom of the SEN. Bubbles inside the SEN are all projected onto the symmetry plane, the bubbles are drawn to scale and are colored by their diameter range. Accumulation of argon bubbles are seen at the top of ports and the swirl center. The back flow pushes argon bubbles into the SEN where they meet the downward going jet and accumulated at the top recirculation zone of the port due to buoyancy force. This implies formation of short-life argon pocket at these spots, although the calculation is not exact, owing to local mass imbalance inherent to the computational method in regions of high gas fraction. In the bottom well, the low pressure region of the swirl contains also many bubbles and coalescence and breakup of bubbles may happen there due to higher shear rate. When the swirls change the rotation directions, these argon bubbles at the bottom of SEN are flushed into the mold region. Also note that when the swirl growth
larger enough to take over the entire port, it moves the bubbles at the top portion of ports to the lower high velocity exiting jet region, from where those bubbles will be sent into the mold region.

Figure 7.10: Contours of y-velocity in the symmetry plane at SEN bottom at (a) \( t = 23.5 \) s and (b) \( t = 24 \) s. All bubbles are projected into the symmetry plane and are drawn to scale.

Figure 7.11 shows contours of \( u_x \) in the center plane and argon bubbles that are within 2 mm distance to the middle plane (-2 \( < y_p < 2 \) mm) at \( t = 24 \) second. The size of the argon bubbles are drawn to scale.

Argon bubbles enter the mold are close to each other, then due to the transient flow and the turbulent effect, argon bubbles are distributed as they were carried by the jet. So, more bubbles are seen close to the jet than in the top region of the mold. The jets are close to the bottom of the two ports with a maximum speed
~1.7 m/s, and back flows are seen in the upper portion of the port. Figure 7.12 zoom in at the region close to the jet, the vectors shows the velocity of argon bubbles. Most bubbles entering the mold with a higher velocity of ~1.2-1.5 m/s following the thin layer of jet which is around 3 cm thick. Near the top backflow region of the port, some argon bubbles travel reversely into the port. Outside port above the jet, an eddy is shed off which also carrying some large bubbles. This shedding of vortex may due to two reasons: 1) the KelvinHelmholtz instability due to the high velocity shear at the jet boundary and 2) the buoyancy force caused by a group of bubbles tear the jet apart which intensified the instability. Close to SEN, the path lines shows the buoyancy of bubbles drives the flow up, then it meets the downward going flow and forms small eddies in the upper portion of the mold. Near the top surface, most bubbles rise biased to the SEN side. However, some small bubbles move downward with the steel flow.

Figure 7.13 shows the distribution of bubbles in the caster at \( t = 24 \) s. To visualize both large and small bubbles, each bubble is represented by a point with color indicating its diameter range. Although the large bubbles \((d_p \geq 2 \text{ mm})\) are occasionally sent deeper into the mold by the fluctuating jets, they mainly rise into the upper portion of the caster owning to larger buoyancy forces. The buoyancy force exerting on the 1 mm bubbles is \(8\times\) smaller than that of the 2 mm bubbles, thus lots of \(\leq 1 \text{ mm}\) bubbles are transported deep into the caster by the liquid steel. Note that, the bubbles in Figure 7.13 are not drawn in scale. Thus, all tiny bubbles, which should not be seen if plot in real scale, are presented as one pixel in the figure. This visualization defect causes much more bubbles are “visible” in the figure.
7.3.3 Capture and removal of argon bubbles

During the simulation, bubbles are continually injected into the domain with an injection rate of 40,700 particles per second. After a few seconds, lots of large bubbles escape from the top surface and some small bubbles are captured by the solid shell. In Figure 7.14, black lines shows the accumulation of argon bubbles inside the caster. It starts with zero and increases linearly in the beginning of the simulation because bubbles have not reached the top surface or being captured yet. After 30 second, the total number of bubbles inside the domain reaches $2.2 \times 10^5$ and increases very slowly due to reach a quasi-steady state where escaping rate
plus capture rate roughly equals the injection rate. The triangle symbols shows the bubble escaping rate (bubbles removed from top surface) is roughly equal to the injection rate of 40,700 particles per second. The capture rate (bubbles are captured by the shell) is $\sim$400-600 bubbles per second which is 100 times smaller than the injection rate, indicating an overall capture fraction of $\sim$1%.

Figure 15 shows the number of bubbles remains in the domain versus bubble size. For small and tiny bubbles ($d_p \leq 0.3$ mm) the remaining bubbles inside the domain is almost proportional to the number of bubbles injected into to domain. However, the number of large bubbles remains in the caster is much less than injected, owing to high escaping rate of large bubbles. The percentile numbers in the figure shows fraction of bubbles remains in the caster, which is the ratio of the square symbols to the circles. The remaining fractions of large bubbles are small (4 to 22%) due to most of them escaped. The remaining fractions of tiny bubbles ($d_d p \leq 0.1$ mm) are $\sim$68%, due to lots of them are captured because they are smaller than the PDAS. A maximum remaining fraction of $\sim$77% is seen for 0.1 and 0.2 mm bubbles, because the buoyancy force of those bubbles are not large enough to easily bring them to the top surface, and they are also more difficult to be captured by the solidifying shell than the tiny bubbles.

7.3.4 Compare capture rate and fraction with previous RANS model

Figure 7.16 shows the capture fraction of different bubbles compare with previous RANS model results.[2] The RANS with random walk model predicts $\sim$85% of tiny bubbles ($d_p \leq 0.1$ mm) are captured, while LES model predict a capture fraction of 20$\sim$35% which is much less. These tiny bubbles are smaller than the PDAS and the capture criterion is generally “touch = capture”. Therefore, the result shows in LES simulation the chance for tiny bubbles to penetrate through the boundary layer and touches the solidification shell is much lower than that predicted by the RANS with random walk model. The random walk model
over predicts the deposition of small bubbles due to its isotropic turbulence assumption which generates a relatively larger velocity fluctuation component normal to the wall. For small bubbles (0.2 ≤ \(d_p\) ≤ 0.3 mm), LES shows the capture fraction is 1% which is much lower than that predicted using RANS. No ≥ 1 mm bubbles are captured during this study.

Figure 7.17 compares the predicted capture rate with previous RANS predictions.[2] LES shows the capture rate of tiny bubbles (\(d_p\) ≤ 0.1 mm) is 75~200 bubbles per second, while previous RANS shows 180~400 bubbles per second. The capture rate of small bubbles (0.2 ≤ \(d_p\) ≤ 0.3 mm) is 10~20 bubbles per second, which is about 50~100× smaller than that predicted by RANS. The RANS simulation predicted that the capture rate of 1 mm bubbles is 10 per second, but in LES simulations no 1 mm bubbles are captured in 33 seconds.

Figure 7.18 shows the predicted average bubble diameter beneath the strand narrow face surface. These results are compared with both RANS prediction and plant measurement.[1, 2] The star symbols shows the
Figure 7.18: Compare predicted average bubble diameter on NF with previous RANS model and measurements.

Results of RANS random walk model with a simple capture criterion which assumes “touch=capture” for all bubbles. It greatly overpredicts the average bubble diameter. After using the advanced capture criterion [15, 16, 18] the results are seen to be improved, but still predicts a larger average bubble diameter. This may be caused by the over-prediction of capture of those small bubbles (0.2 ≤ dp ≤ 0.3 mm) as mentioned earlier. After replacing the steady RANS model with random walk approach with transient LES approach, the predicted average bubble diameter is closer to that measured at Baosteel.[1, 2] Similar results are seen for the wide face inner radius. Figure 7.19 shows the average diameter of bubbles obtained from a sample cut from the center region (center of the wide face inner radius). LES predicted the average bubbles diameter is also ~0.1 mm and matches with the measurement better. However, both models predicted no bubbles are captured at region very close to the top strand. One possible reason is that hooks are not included in these models, but hooks can capture bubbles close to the meniscus.[7]
Figure 7.20 shows bubbles captured by shell during $11 < t < 33s$. These small bubbles can be captured deep into the caster, which agrees with previous studies.[2, 17, 23, 27] There are 3196 and 3672 bubbles captured by the outer and inner radius, respectively. The trend of more bubbles are captured by inner than the outer radius also agreed with previous RANS model[1, 2], although the LES predicts a much smaller capture rate. $\sim$600 bubbles are captured by each of the narrow face which is 5$\sim$6$\times$ less bubbles when compared with that captured by the wideface. This is different from the RANS results which predicts a similar capture rate of narrow face and outer radius of wide face. The slight banding effect is due to the staircase mesh of the shell so should be ignored.[15, 34] No $\geq$ 1 mm bubbles are captured during the simulation. On the inner radius, more 200 and 300 micrometer bubbles are captured close to port exits. Most 100 micrometer bubbles are captured in the upper region of the mold that close to the ports. Less bubbles are captured in the top center region of the caster, because most small bubbles are following the jet and the chance for bubbles enter into the region between the SEN and wide face is low. On wide face, most 100 micrometer bubbles are captured $\sim$0.4 m below the top surface ($\sim$2 cm beneath the strand surface) and $\sim$0.65 m, because the PDAS is larger than the bubbles size.

Figure 7.21 shows the distribution of all the bubbles escaped from the top surface for $11 < t < 33$ s. Larger bubbles ($\geq$3 mm) escapes close to the SEN, but the 1 and 2 mm bubbles escapes at anywhere on the top surface. Both LES and previous RANS model[2] shows $geq$3 mm bubbles leave close to SEN and biased toward the inner radius of the caster, and less bubbles escaping from the region between the SEN and wide faces. However, the RANS model predict the $\geq$3 mm bubbles escapes within $\sim$0.3 m away from the SEN
Figure 7.21: Bubbles escaped from the top surface. Sizes of the bubbles are not drawn to scale.

center (the region enclosed by the dashed line), while the present LES model shows they leave within 0.5 m away from the SEN center. This may due to the RANS model cannot predict the transient oscillation of the jet which sends the bubbles to different locations.

7.3.5 Effects of EMBr on flow field and argon bubble transport

Applying magnetic field affects the time-averaged flow behavior in the caster as well as the transient flow structure in the mold.[43, 44, 47, 48] It reduces the mean surface velocity and fluctuations[20, 41–45, 47, 48]. Figure 7.22 shows the time-averaged flow in SEN, center of mold and 1 cm below top surface after using EMBr. Figure 7.22(a) shows in the upper portion of SEN, the flow field is similar to that without EMBr owing to weak local magnetic field strength. A big counter-clockwise rotating swirl (with downward velocity near outer radius) is seen at the bottom of SEN which is different from two equal sized swirls in the zero
magnetic field case. In both cases, the steel flow first enter the inner radius side of the SEN then deflected from to the outer radius side. With EMBr, the induced Lorentz force reduces the fluctuations inside the SEN and makes the counter-clockwise rotation dominates at the bottom. As shown in Figure 7.22(b), in the mold region, the velocity is much smaller than that without EMBr. The upper rolls are closer to the thinner and shorter jets. Near the ports top, two vortices are formed due to the buoyancy of argon bubbles leaving the two ports. Figure 7.22(c) shows close to the top surface, an outer-radius towards inner-radius cross flow (∼0.1 m/s) appears. The formation of cross flow is caused by the strong counter-clockwise rotating swirls inside the SEN which send most argon bubbles to the outer-radius side of the mold. Figure 7.23 compares the side views of bubbles distribution inside the mold with and without EMBr. As shown in Figure 7.23(a),

![Figure 7.23: Side views of bubbles inside the caster at t = 25 s: (a) EMBr off; (b) EMBr on. Showing all bubbles and bubbles are drawn to scale.](image)

without EMBr, bubbles are sent deeper into the mold with roughly uniform distribution between inner and outer radius. However, after applying EMBr, more bubbles are sent to the outer-radius side of the mold. These bubbles rise closer to the outer-radius side of the mold and induce cross flow near the top surface.

### 7.3.6 Effects of EMBr on argon bubble capture and removal

Figure 7.24 (a) to (c) show the captured location of bubbles on wide face inner-radius, outer-radius and narrow face. Compare without EMBr (Figure 7.20), after applying EMBr bubbles are captured in the upper portion of the mold and less bubbles are captured. Similarly, no ≥ 1 mm bubbles are captured. The total number of captured bubbles is also much less then with EMBr.
EMBr also changes the accumulation of bubbles remain inside the caster. Figure 7.25 shows the effect of EMBr on bubble size population inside the caster at $t = 29$ second. The numbers of different bubbles injected are shown with blue circles which are same in both cases with and without EMBr. For tiny bubbles ($d_p \leq 0.1$ mm), $\sim 70\%$ still remain inside the caster for both cases. After applying EMBr, the number of small to medium sized bubbles ($0.2 \leq d_p \leq 1$ mm) inside the caster are reduced. This is because the Lorentz force changed the flow pattern and suppressed the turbulence and fluctuations, which cause middle sized bubbles hardly move into deeper portion of the mold, and further lead to shorter residence time inside the caster. EMBr has less effect on large bubbles ($2$ mm $\leq d_p$) because buoyancy forces are larger on these bubbles and they mainly float to the top surface in both cases.
The total capture rate of argon bubbles are reduced after using EMBr. Figure 7.26 compares the total capture rate for both cases. With no EMBr applied, the total capture rate fluctuates around 400~600 bubbles per second after $t = 20$ second. With EMBr, the capture rate is reduced by $\sim 2\times$ which fluctuates around 200~300 bubbles per second. It is partially because the jets are shorter, and the drag force on bubbles are larger, thus less bubbles are sent to narrow face region which causes a reduction of number bubbles captured on narrow faces. EMBr reduces the turbulence fluctuations and the slow down the motion of steel inside the caster, which also causes less bubbles move deeper into the caster. And the most bubbles are floating upward and escape at top surface. Bubbles are easier to be captured deeper inside the caster due to larger PDAS, but the use of EMBr reduces the chance of bubbles move deeper into the caster which also reduces the capture rate.

To study the effect of EMBr on the capture of bubbles of different sizes, Figure 27 (a) and (b) compares the capture fractions and rates of different bubbles with and without EMBr. Without EMBr, the capture fraction of tiny bubbles ($d_p \leq 0.08$ mm) is $\sim 35\%$, and turn on the EMBr reduces the capture fraction to $\sim 25\%$. The effect of EMBr on capture fraction of 0.1 mm bubbles is significant. EMBr reduces the capture fraction from $\sim 20\%$ to $\sim 4\%$. This is because these bubbles has larger size which is comparable with the PDAS. The use of EMBr reduces the chance of those bubbles move into deeper region of the caster where the PDAS is large. Thus, with EMBr the capture rate of these bubbles are reduced significantly. The capture fraction of bubbles 0.2 and 0.3 mm bubbles is $\sim 1\%$ and the effect of EMBr on the capture of these bubbles are small. Figure 27 (a) shows when no EMBr, the capture rate of bubbles with $d_p \leq 0.1$ mm are 50~200 bubbles per second. Use of EMBr reduces the capture rate of these tiny bubbles. The capture rate of 0.2 and 0.3 mm bubbles are $\sim 20$ per section and are not affected by the EMBr.
Figure 7.27: Compare the predicted (a) bubble capture fraction and (b) capture rate with/without EMBr

7.4 Conclusions

A multi-GPU-based LES code CUFLOW is applied to study the two-phase steel flow and transport capture of argon bubbles inside a commercial caster with and without the use of double ruler EMBr. The effect of Lorentz force on bubble drag is included by using a modified drag coefficient. The effect of sub-grid-scale on bubbles dispersion is molded. An advanced force balance capture criterion was used to predict the capture of argon bubbles. The important findings of this work are summarized below:

1. With casting speed of 1.5 m/min and 8% of argon gas injection rate, double roll pattern is seen when EMBr is turned off. The buoyancy force of bubbles are not strong enough to cause reverse flow (from SEN to NF) on top surface. This flow pattern is different from previous study which used a RANS approach.

2. Without EMBr, previous RANS approach predicts most large bubbles escaped very close to the SEN (<0.3 m away from SEN center), while LES shows the transient turbulent jets can send larger bubbles further to 0.5 m away from SEN center. This wide distribution of argon bubbles reduces the flotation effect of the bubbles in the near SEN region, thus no reverse flow is seen.

3. This Lagrangian approach predicts the bubbles accumulation in the back flow region of beneath slide gate and in the top of the ports, which agrees with water experiments. Accumulation of argon bubbles are also seen in the low pressure region of vortex center at the bottom of the SEN.

4. Without EMBr, the LES model predicts less bubbles are captured when compare with previous RANS model[2], especially for $0.2 \leq d_p \leq 0.3$ mm bubbles. The capture fraction is only 1% which is much less than that predicted by previous RANS simulation (~60%). The cause of this discrepancy might be (1) the difference flow pattern predicted between the two models; (2) the use of random walk in RANS model which over-predict the chance of bubbles hit the walls.

5. Both LES and previous RANS model predict no large bubbles (>2 mm) are captured. The LES
model predicted the average bubble diameter of argon bubbles is \(\sim 0.1\) mm which matches better with measurements.

6. Without EMBr, the transient LES simulation shows two equal sized swirls are at the bottom of SEN. However, with EMBr, only a counter-clockwise swirl is seen at the bottom of SEN. This dominating swirl causes more bubbles were sent to the outer-radius side of the mold and lead to cross flow near the top surface.

7. EMBr reduces the penetration depth of the two jets exiting from the ports. Less bubbles were sent to the narrow face side, nor sent deep into the caster. This causes less bubbles reduction in the capture fraction and rate of argon bubbles. The total capture rate reduces from 500\(\sim\)600 bubbles per second to 200\(\sim\)300 bubbles per second after use EMBr.

8. The EMBr significantly affects the capture rate and capture fraction of tiny and small bubbles. For tiny bubbles, use EMBr reduces the capture rate by 20 bubbles per second. The capture fraction reduces from \(\sim 35\%\) to \(\sim 25\%\). Use EMBr causes the capture rate of 0.1 mm bubbles drops from 100 to 20 bubbles per second. However, capture rate for 0.2 and 0.3 mm bubbles is 15\(\sim\)20 bubbles per seconds (with a capture fraction \(\sim 1\%\)), and EMBr has less effect on the capture rate and capture fraction of these bubbles.

7.5 References


[8] L. Zhang, J. Aoki, B. G. Thomas, Inclusion removal by bubble flotation in a continuous casting mold,


[41] Z.-D. Qian, Y.-L. Wu, Large eddy simulation of turbulent flow with the effects of DC magnetic field


Chapter 8

Effect of Hooks on Bubble Capture

8.1 Introduction

Previous RANS and LES models used in Chapter 3 and 7 show that the predicted captured bubbles have similar trends as the plant measurements, except in the first layer near to the outer surfaces, where more bubbles are captured than predicted. This might be due to the model neglecting the effects of meniscus hook capture, or assuming immediate escape of bubbles that reach the top surface. In this Chapter, a hook capture mechanism is developed for the RANS model. Then, the model is used to predict the capture of argon bubbles under the same casting conditions as used in Chapter 3. Details of the process parameters are available in table 3.1.

8.2 Hook Capture Mechanism

As shown in Fig. 8.1, a particle/bubble enters into a hook zone has three different possible fates: (1) capture by hook structure; (2) capture by the solidifying shell; (3) escape from hook zone and return to bulk fluid.

![Figure 8.1: Modeled hook shape and the fate of a bubble/particle after it enters hook zone](image-url)
region. Inclusion particles and argon bubbles enter hook zone may not be captured immediately because there is a certain distance between two hooks. This distance $d_{\text{hook}}$ is:

$$d_{\text{hook}} = V_c f_0^{-1}$$  \hspace{1cm} (8.1)

where $f_0$ is the mold oscillation frequency and $V_c$ is the casting speed. As illustrated in Fig. 8.2 If the bubble stays in the region longer than a certain time of $t_{\text{hook}}$, the bubble will be captured by the hook structure. It is important to determine the representative time threshold $t_{\text{hook}}$. In this model, this threshold is approximated by:

$$t_{\text{hook}} = 0.5 f_0^{-1}.$$  \hspace{1cm} (8.2)

This is half of the time required for a hook to travel half of the hook separation distance. When a particle/bubble enters the hook zone, a timing counter is activated and the time of the particle/bubble remained in the hook zone is recorded as $t_c$. The particle/bubble is captured by the hook if $t_c > t_{\text{hook}}$ and the current location of the particle/bubble is projected vertically onto the shell as illustrated in Fig. 8.2(a). This capture mechanism is independent with the force capture criterion and this hook capture criterion is checked in each particle timestep. As shown in Fig. 8.2(b), the counter $t_c$ is reset to 0 if the particle/bubble in not in the hook zone (pink region in Fig. 8.2(b)). In Fig. 8.2(b), $t_1$ is the time taken for a bubble travel on the yellow path, and at $t_c = t_1$ the bubble escaped from the hook zone and therefore $t_c$ is set to 0. Later, the bubble enters the hook zone again and travels along the green path, and $t_c$ starts accumulate since it re-enters the hook zone. Finally $t_c > t_{\text{hook}}$ and the bubble is captured by the hook. The above hook capture mechanism was implement into Fluent using UDF. Two simulations with different hook depths, 6 mm and 3 mm, were carried out with same oscillation frequency of 25 Hz ($t_{\text{hook}} = 0.25 s$). The quasi-steady state flow
field used in Chapter 3 was used, and the details of the flow field can be found in Section 3.4.1.

8.3 Results and Discussion

8.3.1 Bubbles Captured Using 6 mm Hook Depth

To examine the hook capture mechanism, locations of the bubbles captured by 6 mm hooks were shown in Fig. 8.3 and Fig. 8.4. Due to an unsymmetrical flow field, the number of bubbles captured by inner and outer radius hooks are also different. Figure 8.3(a) and (b) show the captured small bubbles and large bubbles by the inner radius hooks. Each point (bubbles are not drawn to scale) in these figures represents a hook captured bubble, different colors are used to indicate different size of the bubbles. There are 229 bubbles captured by the inner radius hooks, including 148 $d_p \leq 300\mu m$ bubbles and 81 $d_p > 300\mu m$ bubbles. All the bubbles are captured close to the meniscus.

Figure 8.4 shows the bubbles captured by the outer-radius hooks. 1067 argon bubbles were caught during the 6 seconds, including 153 $d_p \leq 300\mu m$ bubbles and 914 $d_p > 300\mu m$ bubbles. With a 6 mm hook depth,
both inner and other radius hooks capture \(\sim 150\) small bubble \((d_p \leq 300\mu m)\). However, \(10\times\) more large bubbles \((d_p > 300\mu m)\) were captured by the outer radius. This is because hook zone is really close to the shell \(<6\) mm in this case), and small bubbles entering the hook zone are easier to be captured by the shell before they are captured by the hook mechanism. However, larger bubbles are harder to be captured by the shell. After being rejected by the force balance capture criterion, they still remains inside the hook zone. After remaining in the hook zone longer than the specified \(t_{\text{hook}}\), those large bubbles are captured by the hooks.

### 8.3.2 Bubbles Captured Using 3 mm Hook Depth

Figure 8.5 and 8.6 shows the location of captured bubbles by the inner radius hooks and outer radius hooks with a hook depth of 3 mm. After using a shorter hook depth (3 mm), the hook affected zone becomes much smaller. Thus, the captured bubbles are much more close to the meniscus. As shown in Fig. 8.5(a), only 28 small bubbles are captured by the inner-radius hook and this is \(\sim 5\times\) smaller than the case with 6 mm hooks. No larger bubbles \((d_p \leq 300\mu m)\) are captured by 3 mm hooks. The bubbles captured by the outer radius 3 mm hooks are also significantly less than that captured by the 6 mm hooks. There are only \(48\) \(d_p \leq 300\mu m\) bubbles and \(6\) \(d_p > 300\mu m\) captured by the outer radius 3 mm hooks, which is \(\sim 3\times\) and \(\sim 150\times\) less than captured by outer radius 6 mm hooks.

### 8.3.3 Capture Locations and Number of Small Bubbles

The captured locations of the small \((d_p < 0.1mm)\) and medium-sized \((0.1 \leq d_p \leq 0.3mm)\) bubbles are shown in Fig. 8.7 and Fig. 8.8, for using 3 mm and 6 mm hook depth, respectively. The horizontal black lines show the location of the sample surfaces examined in the measurements. All figures show that the number of bubbles captured by the solidifying shell decreases with distance down the caster. However, many
Figure 8.6: (a) Small bubbles captured by outer-radius hook (hook depth 3 mm); (b) Large bubbles captured by outer-radius hook (hook depth 3 mm). Legend shows the bubble diameter (µm).

Figure 8.7: Small bubbles captured by WF-IR (left), WF-OR (middle) and NF (right) using 3 mm hook depth.
small bubbles are captured deep in the caster, which agrees with previous studies in Chapter 3. Comparing with the predicted captured number of bubbles by the shell using the advanced capture criterion without hook capture mechanism, the use of hook capture mechanism predicts more bubbles to be captured near the meniscus (strand surface). With a 3 mm hook depth, model predicted 3% more bubbles captured by shell. With 6 mm hook, the model predicted 10% more bubbles captured on WF-OR and NF.

8.4 Compare with Previous Prediction and Plant Measurements

The predicted number of argon bubbles by the solidifying shell and their average diameter are compared with previous predictions without hook capture mechanism and plant measurements(See Chapter 3). Figure 8.9(a) and (b) show the predicted and measured number of bubbles captured per $150 \times 150 \text{ mm}^2$ sample layer of the center sample and the quarter sample, respectively. Details of the sample locations and the measurements are available in Section 3.5. In the center sample, all of the models under-predict the number of captured bubbles close to the meniscus region, which means captured close to the strand surface. Hook mechanism predicts slightly more bubbles captured near the second sample layer ($\sim 6 \text{ mm beneath strand surface}$). As shown in Fig. 8.9(c), all models also under predict the number of captured bubbles near meniscus. Figure 8.10(a) and (b) show the predicted and measured average bubble diameter (true diameters) on each examined sample layers of the center and quarter region samples. Increasing hook depth causes larger average diameter in first
Figure 8.9: Predicted number of bubbles captured per sample layer at (a) inner radius center region, (b) inner radius quarter region and (c) NF region

2 layers due to more large bubbles captured. Using a 6 mm hook depth, the model significantly over-predicts

Figure 8.10: Predicted average bubble diameter in each sample layer at (a) inner radius center region, (b) inner radius quarter region and (c) NF region sample

the average bubble diameter near the meniscus in both the quarter region and narrow face samples. This is caused by lots of large bubbles were predicted to be captured by the hooks. With a 3 mm hook depth, the predicted number of average bubble diameter in the quarter region sample is also higher. However, this is causes by a big 1 mm bubble is predicted to be captured (see Fig. 8.6(b)). This single large bubble causes a huge increase in the mean diameter.

8.5 Conclusions

Hook capture mechanism were added into the model and used with advanced capture criterion to predict particle transport and capture. The model predicted hook captured bubbles were close to meniscus region. With 3mm hook depth, model predicted 3% more bubbles captured by shell; with 6mm hook, predicted 10% more bubbles captured on WF-OR and NF. With 6 mm hook depth, the model predicted more large bubbles captured by hook which causes larger average bubble diameter. Effect of hook is not the main reason for predicting less bubbles captured at region close to meniscus.
Chapter 9

Conclusions and Recommendations

This research studied the motion of liquid steel, transport and captures of argon bubbles inside continuous casting of steel caster. To mathematically model the complex continuous casting process, an efficient Reynolds-Averaged Navier-Stokes based computational model is developed using the commercial package ANSYS FLUENT. An advanced capture criterion is implemented using User Defined Functions. The model is validated by comparing with plant measurements. Then a more accurate LES model is developed to further understand the transient phenomena. In the LES model, a multi-GPU version of CUFLOW is developed to solve the incompressible Navier-Stokes equations. A two-way coupled Eulerian-Lagrangian discrete particle/bubble transport module is implemented. A magnetohydrodynamics module is developed to investigate the effects of the EMBr system on the fluid flow and argon bubbles transport. A volume of fluid based method is used to study argon bubble rise in liquid steel pool under an external magnetic field, and a modified drag coefficient is extrapolated from the results. The modified drag coefficient is used in the Eulerian-Lagrangian calculations to include the effect of EMBr on argon bubble motion. A new model of hook capture mechanism is developed to understand the effect of hooks on argon the capture of argon bubbles close to the meniscus. Many numerical simulations are conducted to understand the effects of EMBr strength on the fluid flow in the caster, the effects on transport and capture of argon bubbles. The important conclusions are summarized below:

9.1 Effects of Submergence Depth

(1) The results confirmed that the change of submergence depth has almost no effect on the flow inside the SEN or the swirls inside the port.

(2) With no argon injection, both 170 mm and 200 mm submergence depth produces double roll patterns inside the mold cavity. Both submergences can causes surface velocity greater than 0.4 m/s which may lead to slag entrainment. Increasing the submergence depth by 17 pct (from 160 to 200 mm) causes a 15 pct decrease in top surface mean flow velocity. However, the surface level fluctuations are less affected by this
Effects of Casting Speed

1. Casting speed affects the swirls inside the SEN ports. With a lower casting speed of 1.3 m/min, a counterclockwise rotation swirl shows at the bottom of the SEN. A higher casting speed of 1.8 m/min generates two symmetric swirls in each port.

2. Higher casting speed leads to higher surface velocity. With a casting speed of 1.5 m/min, RANS simulations show the top surface velocity remains in the safe window of 0.1 to 0.3 m/s, which agrees with the mean flow velocity predicted using LES models. However, the LES models show the transient flow velocity can be much larger than 0.4 m/s.

3. With higher casting speed, the jet is stronger at the bottom of the ports with a larger backflow region present at the top of ports. With lower casting speed, the backflow region is smaller near the port top, but stronger at the swirl center. This is because the two symmetrical swirls formed under high casting speed are weaker than the big swirl formed under the low casting speed.

Effects of EMBr on Steel Flow when Argon Injection Can Be Ignored

1. EMBr causes flow in the nozzle to be more uniform, reducing the extent of asymmetric flow caused by the slide gate. It also increases downward velocity and momentum along the SEN walls, forming an M-shaped velocity profile in the lower part of the SEN.

2. Swirl in the nozzle bottom decreases with increasing casting speed (via the accompanying increase in slide gate opening) and with increasing EMBr field strength across the nozzle.

3. Backflow is often seen in the top portion of the port, or in the core of the big swirl exiting the port. Increasing casting speed from 1.3 m/min to 1.8 m/min has little effect on the size of the backflow region. EMBr makes the upper backflow region larger (occupying more than 1/3 of the port).

4. With EMBr, the jets leaving the ports become thinner and stronger. This is affected by the M-shaped profile inside the SEN and the swirl in the nozzle bottom.

5. When the slide gate moves towards the inner radius side of the nozzle, the time averaged velocity near the outer radius is higher, although the flow direction is straight towards the SEN without cross-flow. Close to the SEN, although the mean velocity is small (<0.1 m/s), the transient results have high velocity (~0.4
m/s) regions close to the nozzle. With EMBr, the top surface velocity is more uniform with no velocity bias to either side.

(6) Without EMBr, vortices form in the corner region close to the NF wall which rotate opposite to the main top recirculation region. These vortices are suppressed with EMBr.

(7) Increasing casting speed from 1.5 to 1.8 m/min without EMBr, causes the top surface velocity to exceed 0.4 m/s, the surface profile variations to exceed 10 mm, and surface level fluctuations of $\sim13$ mm, which may cause slag entrainment. These variations are all lowered with EMBr.

(8) However, with both EMBr rulers operating in the caster of this study, holding the meniscus level at middle of the top ruler of the EMBr greatly reduces the top surface velocity (to $\sim0.05$ m/s), which may cause inadequate heat transfer at meniscus and may lead to meniscus freezing and slag entrapment.

(9) EMBr modifies the top surface level profile from a “W” shape to a “U” shape, lessens its variations, and reduces the turbulence kinetic energy at the top surface.

(10) For best steel quality, it is recommended to operate this caster at 1.8 m/min with only the bottom EMBr ruler turned on at 850 A.

9.4 Effects of EMBr on Argon Bubble Rise in Liquid Steel

(1) The external magnetic field reduces the rise velocity of argon bubbles, suppresses bubble shape oscillations and leads to a more rectilinear path.

(2) For a 3 mm argon bubble rising under a magnetic field of 0.2 T, the terminal rise velocity is seen to slightly increase compared with that of the no magnetic field case. This is caused by the fact that the rising path of the bubble is more rectilinear. Further increase of the magnetic field reduces the rise velocity.

(3) When rise in liquid steel without EMBr, a 7 mm bubble rise undergoes large deformations and shape oscillations. The oscillatory bubble shape further affects its rise velocity as well as the wake behind. During the oscillations, the bubble rises slower with a larger cross section area.

(4) Shape oscillations of rising argon bubbles generate hairpin structures in the wake. Without the magnetic field, the shape of a 7 mm bubble oscillates at a frequency around 28 Hz. With a large magnetic field strength, the unsteady shape deformations are suppressed and the hairpin type complex wake structures behind the bubble almost disappear.

(5) Although the gas is almost not electrically conducting, by modifying the fluid velocity adjacent to the bubble, the applied magnetic field reduces the circulation inside the bubble.

(6) With applied magnetic field, the argon bubble is compressed in the direction perpendicular to the
applied magnetic field direction. The observed results are a direct result of the Lorentz force distribution which increases the resistance to flow motion in the plane perpendicular to the magnetic field.

(7) A correlation of $C_D/C_{D0}$ with the Stuart number are extrapolated from the simulations.

9.5 Effects of Capture Criterion on Model Prediction

In the work, two capture criteria were implemented into the RANS model. The advanced capture criterion showed good agreement with measurements of the number, locations, and sizes of captured bubbles, especially for larger bubbles. The relative capture fraction of large (1 mm) bubbles of 0.3% was close to the measured 0.2%, and occurred very near the top surface. Some important findings are summarized below:

(1) The slide gate does not cause biased cross-flow at the top surface for single phase flow. However, when argon gas is injected with the sliding gate moving towards the IR, more Ar bubbles leave the ports towards the IR and lead to cross flow on the top surface from IR to OR, and corresponding non-uniform bubble capture on the IR and OR;

(2) Almost 85% of small ($< 0.08 \text{ mm}$) bubbles are captured. A very small fraction of large bubbles is captured ($< 0.02\%$). This fraction drops almost one order of magnitude with increasing bubble size from 1 to 2 mm and from 2 to 3 mm;

(3) The removal fraction of large bubbles ($1 \leq d_p \leq 5 \text{ mm}$) was 99.98\%, for medium bubbles ($0.1 \leq d_p \leq 0.3 \text{ mm}$) was around 48.5\% and for small bubbles ($d_p < 0.1 \text{ mm}$) was close to 15\%;

(4) Most bubbles are captured very near to the meniscus. Deeper than 0.5 m below the meniscus, the capture fraction of medium bubbles is about half that of small bubbles;

(5) The predicted bubbles captured shows similar trends as the plant measurements, except in the first layer near to the outer surfaces, where more bubbles are captured than predicted. This might be due to the model neglecting the effects of meniscus hook capture, or assuming immediate escape of bubbles that reach the top surface.

9.6 Effects of EMBr on the Capture of Argon Bubbles

(1) With casting speed of 1.5 m/min and 8\% of argon gas injection rate, double roll pattern is seen when EMBr is turned off. The buoyancy force of bubbles are not strong enough to cause a reverse flow (from SEN to NF) near the surface. This flow pattern is different from previous study which used a RANS approach.

(2) Without EMBr, previous RANS approach predicts most large bubbles escaped very close to the SEN ($< 0.3 \text{ m away from SEN center}$), while LES shows the transient turbulent jets can send larger bubbles
further to 0.5 m away from SEN center. This wide distribution of argon bubbles reduces the flotation effect of the bubbles in the near SEN region, thus no reverse flow is seen.

(3) Eulerian-Lagrangian approach predicts the bubbles accumulation in the back flow region beneath slide gate, near the top corner of the ports, and the low pressure swirls center. These findings agrees with water experiments.

(4) Without EMBr, the LES model predicts less bubbles are captured when compare with previous RANS model, especially for $0.2 \leq d_p \leq 0.3$ mm bubbles. The capture fraction is only 1% which is much less than that predicted by previous RANS simulation ($\sim$60%). The cause of this discrepancy might be (1) the difference flow pattern predicted between the two models; (2) the use of random walk in RANS model which over-predict the chance of bubbles hit the walls.

(5) Both LES and previous RANS model predict no large bubbles ($>2$mm) are captured. The LES model predicted the average bubble diameter of argon bubbles is $\sim$0.1 mm which matches better with measurements.

(6) Without EMBr, the transient LES simulation shows two equal sized swirls are at the bottom of SEN. However, with EMBr, only a counter-clockwise swirl is seen at the bottom of SEN. This dominating swirl causes more bubbles were sent to the outer-radius side of the mold and lead to cross flow near the top surface.

(7) EMBr reduces the penetration depth of the two jets exiting from the ports. Less bubbles were sent to the narrow face side, nor sent deep into the caster. This causes less bubbles reduction in the capture fraction and rate of argon bubbles. The total capture rate reduces from 500$\sim$600 bubbles per second to 200$\sim$300 bubbles per second after use EMBr.

(8) The EMBr significantly affects the capture rate and capture fraction of tiny and small bubbles. For tiny bubbles, use EMBr reduces the capture rate by 20 bubbles per second. The capture fraction reduces from $\sim$35% to $\sim$25%. Use EMBr causes the capture rate of 0.1 mm bubbles drops from 100 to 20 bubbles per second. However, capture rate for 0.2 and 0.3 mm bubbles is 15$\sim$20 bubbles per seconds (with a capture fraction $\sim$1%), and EMBr has less effect on the capture rate and capture fraction of these bubbles.

### 9.7 Effect of Hook on the Capture of Argon Bubbles

Hook capture mechanism were added into the model and used with advanced capture criterion to predict particle transport and capture. The model predicted hook captured bubbles were close to meniscus region. With 3mm hook depth, model predicted 3% more bubbles captured by shell; with 6mm hook, predicted 10%
more bubbles captured on WF-OR and NF. With 6 mm hook depth, the model predicted more large bubbles captured by hook which causes larger average bubble diameter. Effect of hook is not the main reason for predicting less bubbles captured at region close to meniscus.