Modelling of Oxygen Injection and Splashing in Steelmaking

A Thesis Submitted for the Degree of Doctor of Philosophy

By

Md Morshed Alam

Faculty of Engineering and Industrial Sciences
Swinburne University of Technology
Melbourne, Australia
2012
Declaration

The candidate hereby declares that the work in this thesis is that of candidate alone. It contains no material, which has been accepted for the award of any other degree or diploma at any university and to the best of my knowledge and belief contains no material previously published or written by another person or persons except where due reference is made.

Md Morshed Alam

Certification

This is to certify that the above statement made by the candidate is correct to the best of our knowledge.

Dr. Jamal Naser

Prof. Geoffrey Brooks
To My Loving Parents
Abstract

Supersonic oxygen jet is used in both Basic Oxygen Furnace (BOF) and Electric Arc Furnace (EAF) steelmaking to remove impurities from the liquid iron by oxidation reactions. In EAF steelmaking, the oxygen jet is injected between 35 to 45 degrees from the vertical position whereas in BOF steelmaking the oxygen jet is injected through a vertical lance from the top of the furnace. The understanding of the supersonic jet behaviour, jet-liquid interaction and droplet generation rate from the liquid surface inside the furnace is important in optimizing the steelmaking process.

In the present study, the Computational Fluid Dynamics (CFD) technique and physical modelling was used to study the supersonic jet behaviour, jet-liquid interactions and splashing phenomenon inside the steelmaking furnace. A CFD model was developed first to investigate the effect of high ambient temperature on the supersonic jet because in steelmaking the ambient temperature is between 1700 to 1800 K. The results, when compared with available experimental data, showed that the $k-\varepsilon$ turbulence model, with compressibility terms, underpredicts the potential core length of the supersonic jet at high ambient temperature. A temperature corrected $k-\varepsilon$ turbulence model was proposed in this study that decreases the value of $C_{\mu}$ at higher temperature gradient which in turn reduces the turbulent viscosity. As a result, the growth rate of turbulence mixing region decreases at higher ambient temperatures which increases the potential core length of the jet further. The results obtained by using the modified turbulence model were found to be in good agreement with the experimental data. The CFD simulation showed that the potential core length at steelmaking temperature is approximately 2.5 times longer compared to that at room ambient temperature.

It was then followed by the CFD modelling of coherent supersonic jet where the supersonic jet is shrouded by a combustion flame which results in longer potential core length for the coherent supersonic jet compared with normal supersonic jet. The combustion flame was created using CH$_4$ and O$_2$ as fuel and oxidant respectively. The CFD model showed that the shrouding combustion flame reduces the turbulent shear stress magnitude in the shear layer. As a result, the potential core length of the jet increases. The potential core length of the coherent supersonic jet was also found to depend on the gas
used as the central supersonic jet. The numerical results showed that the potential core length of the coherent supersonic oxygen and nitrogen jet is more than four times and three times longer, respectively, than that without flame shrouding, which were in good agreement with the experimental data.

After modelling the gas phase, another CFD model was developed to simulate the jet liquid interaction. A new approach was proposed where two computational domains were used to avoid the difficulties that arise from the simultaneous solution of compressible supersonic gas phase and incompressible liquid phase. The effect of shrouding gas flow rates on the axial jet velocity distribution, depth of penetration and velocity distribution of liquid phase were investigated. In this case, only compressed air was used to shroud the main supersonic jet instead of a combustion flame. A higher shrouding gas flow rate was found to increase the potential core length, depth of penetration and liquid free surface velocity which in turn contributes in reducing the mixing time. The CFD model successfully predicted the formation of surface waves inside the cavity and consequent liquid fingers from the edge of the cavity which were experimentally observed by the previous researchers.

An experimental study was carried out to investigate the effect of different operating conditions (lance angles, lance heights and flow rates) on the wall splashing rate. Air was injected on water surface in a 1/3-scale thin slice model of the steelmaking furnace where dynamic similarity was maintained using dimensionless Modified Froude number. The splashing rate was found to increase with the increase of lance angle from the vertical and flow rate. The critical depth of penetration as well as the impact velocity for the onset of splashing was found to decreases with the increase of lance angle from the vertical. The dimensionless Blowing number ($N_B$), which is a measure of droplets generation rate, was found to increase with lance angle if the axial lance height is kept constant. It was concluded that the Blowing number theory fails if the cavity operates in penetrating mode. The cavity surface area was predicted to be the most important factor in the generation of droplets at lower lance height but after a certain lance height the jet momentum becomes the dominant factor.

Finally, effort was made to quantify the droplet generation rate from CFD model and validate against the present experimental study. It was found that the modelling of
splashing rate using the Eulerian approach requires a very fine mesh to capture the fine droplets which are computationally very intensive.

The CFD models, developed in the present study, can be used to predict the jet behaviour and impact on the liquid melt inside an industrial furnace at different nozzle inlet conditions. It can also predict the cavity dimensions and oscillations with reasonable accuracy. Thus, the models can be used as predictive tools in the industry. The knowledge of impact velocity and cavity dimensions is important to model the droplet generation rate from the cavity. Modelling of droplet generation is still a significant challenge for CFD but the results from this study indicated that Blowing number theory cannot be used to quantify splashing in the penetrating mode and a further modification is required by including the cavity dimensions as a parameter.
Acknowledgements

PhD is an exciting journey but also becomes very challenging at times. I am grateful to a number of people who in different ways, academically, professionally and psychologically, contributed to my journey. Without their kind contributions and help, it would have been impossible on my part to complete this project successfully within scheduled time.

First of all I gratefully acknowledge my supervisor Dr. Jamal Naser, who arranged everything for my scholarship and gave me the opportunity to work in this project. In particular, I appreciate his invaluable and enterprising guidance throughout the execution of this project.

I would like to express my sincere gratitude to my co-supervisor Professor Geoffrey Brooks for his guidance and valuable suggestions during the progress of this work. I consider myself fortunate that I have got the opportunity to work with my supervisors. Both of them were always accessible and willing to help their students with their research. I have learned research culture and been well trained by my supervisors from the beginning to the final level of my degree.

I am grateful to Dr. Gordon Irons for his excellent supervising during the experimental part of my project and providing me the opportunity to carry out my experimental study at Steel Research Centre of McMaster University, Canada. I appreciate Owen Kelly’s help for the preparation of the experimental set up and his valuable advice to expedite the work as I was there for a short period of time.

I am also grateful to Mr Andrea Fontana of OneSteel mill Laverton, Australia for his valuable discussions and financial contributions especially for the experimental part of the project. I also would like to thank other members of OneSteel mill for their contributions with the industrial data used in this study.

Thanks to all my colleagues of the High Temperature Research Processing Group for their useful advice, ideas and ongoing support. These current and former group members are Nesihan Dogan, Bernard Xu, Nazmul Huda, Winny Wulandari, Behrooz Fateh, Reiza Mukhlis, Abdul Khaliq, Abdus Sattar and Saiful Islam. Specially, I would
like to thank Neslihan, Nazmul and Winny for all those wonderful discussions, sharing of knowledge and experiences during the course of my PhD.

I wish to thank all of my friends outside my Department for their support and understanding over the past four years. Thanks to Mr. and Mrs. Tarek, Mr. and Mrs. Aziz for all those lovely family moments who never let me feel that I am thousand miles away from my home. Many thanks to my long time housemate Anisuzzaman for the delicious cooking and all the fun we had in the last four years.

Above all, this thesis is dedicated to my beloved Mum, Dad and my brothers, Kamrul and Wasif. They have sacrificed many things for me to stand where I am standing now. Their unconditional love and continual support are the keys to all my achievements till now.

Finally, thanks to all of my friends, who don’t know they helped and inspired me, but they did. I am sure I have forgotten someone. I assure you that this is a shortcoming on my part and not on yours. I beg you to forgive me for my oversight.
# Table of contents

ABSTRACT ................................................................................................................................. IV

ACKNOWLEDGEMENTS ........................................................................................................ VII

TABLE OF CONTENTS........................................................................................................... IX

LIST OF FIGURES ................................................................................................................. XIII

LIST OF TABLES .................................................................................................................... XX

NOMENCLATURE .................................................................................................................... XXI

CHAPTER 1 .............................................................................................................................. 1

1. INTRODUCTION ............................................................................................................. 1

1.1 Background ................................................................................................................... 2

1.2 Aim and Scope of This Study ..................................................................................... 5

1.3 Thesis Overview .......................................................................................................... 6

1.4 Publications From This Research ............................................................................. 7

1.4.1 Journals ..................................................................................................................... 7

1.4.2 Conferences .............................................................................................................. 7

CHAPTER 2 .......................................................................................................................... 9

2. LITERATURE REVIEW .................................................................................................... 9

2.1 Overview of Steelmaking Processes ........................................................................... 10

2.1.1 The Oxygen Steelmaking Process ....................................................................... 10

2.1.2 Electric Arc Furnace Steelmaking Process ............................................................ 16

2.2 Theory of Supersonic Gas Jet .................................................................................... 20

2.2.1 Coherent Supersonic Jet ....................................................................................... 27

2.3 Modelling of Impinging Gas Jets on Liquid Surfaces .............................................. 30

2.3.1 Experimental Modelling ....................................................................................... 31
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.3.2 Numerical Approach</td>
<td>48</td>
</tr>
<tr>
<td>2.4 Research Objectives</td>
<td>57</td>
</tr>
<tr>
<td>2.5 Brief Overview of Computational Fluid Dynamics Modelling</td>
<td>58</td>
</tr>
<tr>
<td>2.5.1 Discretization Method</td>
<td>60</td>
</tr>
<tr>
<td>CHAPTER 3</td>
<td>62</td>
</tr>
<tr>
<td>3. NUMERICAL MODELLING OF SUPersonic JET</td>
<td>62</td>
</tr>
<tr>
<td>3.1 Introduction</td>
<td>64</td>
</tr>
<tr>
<td>3.2 Governing Equations</td>
<td>64</td>
</tr>
<tr>
<td>3.3 Turbulence Modelling</td>
<td>67</td>
</tr>
<tr>
<td>3.3.1 Direct Numerical Simulation</td>
<td>68</td>
</tr>
<tr>
<td>3.3.2 Large Eddy Simulation (LES)</td>
<td>68</td>
</tr>
<tr>
<td>3.3.3 Reynolds-Averaged Navier-Stokes (RANS) Model</td>
<td>68</td>
</tr>
<tr>
<td>3.3.4 Turbulence Model Used in The Present Study</td>
<td>70</td>
</tr>
<tr>
<td>3.4 Modelling Of Supersonic Free Jet</td>
<td>75</td>
</tr>
<tr>
<td>3.4.1 Computational Domain</td>
<td>76</td>
</tr>
<tr>
<td>3.4.2 Boundary Conditions</td>
<td>76</td>
</tr>
<tr>
<td>3.4.3 Discretization Schemes</td>
<td>77</td>
</tr>
<tr>
<td>3.4.4 Computational Procedure</td>
<td>78</td>
</tr>
<tr>
<td>3.4.5 Results And Discussions</td>
<td>80</td>
</tr>
<tr>
<td>3.4.6 Conclusions</td>
<td>97</td>
</tr>
<tr>
<td>3.5 Modelling Of Supersonic Coherent Jet</td>
<td>99</td>
</tr>
<tr>
<td>3.5.1 Combustion Modelling</td>
<td>99</td>
</tr>
<tr>
<td>3.5.2 Radiation Modelling</td>
<td>102</td>
</tr>
<tr>
<td>3.5.3 Computational Domain</td>
<td>102</td>
</tr>
<tr>
<td>3.5.4 Boundary Conditions</td>
<td>103</td>
</tr>
<tr>
<td>3.5.5 Computational Procedure</td>
<td>105</td>
</tr>
<tr>
<td>3.5.6 Results and Discussions</td>
<td>107</td>
</tr>
<tr>
<td>3.5.7 Conclusions</td>
<td>123</td>
</tr>
<tr>
<td>CHAPTER 4</td>
<td>125</td>
</tr>
<tr>
<td>4. MODELLING OF TOP JET-LIQUID INTERACTION</td>
<td>125</td>
</tr>
<tr>
<td>4.1 Introduction</td>
<td>126</td>
</tr>
</tbody>
</table>
4.2 Computational Model ................................................................................... 126
4.2.1 Assumptions .............................................................................................. 126
4.2.2 Free Surface Modelling .............................................................................. 127
4.2.3 Governing Equations of VOF Model ........................................................... 129
4.2.4 Turbulent Flow Modelling .......................................................................... 133
4.2.5 Computational Domain .............................................................................. 134
4.2.6 Boundary Conditions ................................................................................. 136
4.2.7 Computational Procedure .......................................................................... 138

4.3 Results and Discussions ................................................................................ 139
4.3.1 Shrouding Gas Jet ...................................................................................... 139
4.3.2 Gas Jet Penetration ................................................................................... 142
4.3.3 Grid Sensitivity Test ................................................................................... 144
4.3.4 Droplets Generation Study ........................................................................ 145
4.3.5 Limitation of Blowing Number ($N_b$) Theory ............................................ 148
4.3.6 Fluid Flow in Liquid Bath ............................................................................ 152
4.3.7 Effect of Liquid Density .............................................................................. 154

4.4 Conclusions ................................................................................................... 155

CHAPTER 5 ..................................................................................................................... 156

5. EXPERIMENT ON INCLINED JETTING AND SPLASHING ......................... 156

5.1 Introduction .................................................................................................. 157

5.2 Experimental Apparatus and Set-up. ............................................................ 157

5.3 Experimental Procedure ............................................................................... 161

5.4 Results and Discussions ................................................................................ 163
5.4.1 Effect of Lance Gas Flow Rates ................................................................. 163
5.4.2 Effect of Lance Angle ............................................................................... 166
5.4.3 Effect of Lance Height ............................................................................... 168
5.4.4 Cavity Oscillations ................................................................................... 169
5.4.5 Splashing Distribution on The Wall ............................................................ 172
5.4.6 Dimensionless Relationship of Depth of Penetration ............................. 174
5.4.7 Critical Depth of Penetration ................................................................... 176
5.4.8 Effect Of Lance Angle on Blowing Number ............................................. 180
5.4.9 Modelling of Full-Scale Furnace ............................................................... 184

5.5 Conclusions ................................................................................................... 186
CHAPTER 6 ..................................................................................................................... 188

6. MODELLING OF INCLINED JET-LIQUID INTERACTION ........................................... 188

6.1 Introduction........................................................................................................ 189

6.2 Governing Equations ...................................................................................... 189

6.3 Computational Domain .................................................................................. 189

6.4 Boundary Conditions .................................................................................... 191

6.5 Computational Procedure............................................................................. 192

6.6 Results and Discussions ................................................................................ 192

6.6.1 Gas Jet Penetration ................................................................................ 192

6.6.2 Grid Sensitivity Test ................................................................................ 195

6.6.3 Cavity Shape and Oscillation .................................................................. 196

6.6.4 Prediction of $\eta$ Value With Lance Angles ......................................... 198

6.6.5 Modelling of Wall Splashing Rate ............................................................ 200

6.7 Conclusions..................................................................................................... 201

CHAPTER 7 ..................................................................................................................... 203

7. CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE STUDY .............. 203

7.1 Conclusions .................................................................................................... 204

7.2 Contribution to New Knowledge ................................................................. 207

7.3 Recommendations for Further Study .......................................................... 208

REFERENCES .................................................................................................................. 210

APPENDIX A.................................................................................................................... 218

APPENDIX B.................................................................................................................... 221

APPENDIX C.................................................................................................................... 225
List of Figures

Figure 1-1 The world annual steel productions from 1970 to 2010 [1]................................. 2
Figure 2-1 Methods of introducing oxygen into steelmaking converter (a) Top blown (b) Bottom blowing (c) Combined blowing [2] ........................................................................ 11
Figure 2-2 Schematics of operational steps in BOF steelmaking process [2]..................... 13
Figure 2-3 Change in melt composition during the blow [3]............................................ 14
Figure 2-4 Production of steel through different steelmaking processes. (Courtesy of GrafTech and OneSteel) .............................................................................................. 16
Figure 2-5 Schematic of an electric arc furnace[10]............................................................. 17
Figure 2-6 Effect of slag foaming on arc radiation [2]...................................................... 20
Figure 2-7 Schematic of a Laval Nozzle with Pressure and Mach number profile[11]. 22
Figure 2-8 Regions of a supersonic jet exiting from a Laval nozzle. ......................... 25
Figure 2-9 Schematics of a (a) conventional and (b) coherent supersonic jet [23]....... 27
Figure 2-10 (a) Cross-sectional and (b) front view of a coherent nozzle [25].............. 28
Figure 2-11 Cavity shape for (a) Top and (b) inclined jetting[37]. ............................... 30
Figure 2-12 Schematic diagram of top jet impingement on a liquid surface. ............... 32
Figure 2-13 Schematic diagram of inclined gas jet impinging on a liquid surface. ...... 36
Figure 2-14 Splashing of liquid due to (a) inclined jetting and (b) top jetting [37]. ....... 39
Figure 2-15 Mechanism of harmonic wave propagation (a) shows formation of the wave and (b) shows the movement of the wave [5]. ......................................................... 40
Figure 2-16 Perspex tray for measuring radial distribution of liquid splashed just above the bath surface [51]............................................................................................ 42
Figure 2-17 Influence of jet momentum on radial distribution of droplets [51]. ......... 43
Figure 2-18 Variation of droplet generation rate with lance heights.[58] ....................... 43
Figure 2-19 Comparison of the results from two modelling experiments [6]..............45
Figure 2-20 Schematic of unstable interface between two moving fluid layer [50]. ....45
Figure 2-21 Effect of Blowing number on the generation rate of metal droplets [7]. ....47
Figure 2-22 Schematics of indentation profile and force balance [63]....................49
Figure 2-23 The computed turbulent kinetic energy profile(cm$^2$/s$^2$) [65]. ............51
Figure 2-24 Surface profile variations with time [75].................................................53
Figure 2-25 Variation in mean depth of penetration with changing lance heights and
flow rates [75]..............................................................................................................53
Figure 2-26 Variation of depth of penetration with flow rates[53]..............................55
Figure 2-27 Variation of depth of penetration with lance angles[53]...........................55
Figure 3-1 Typical point velocity measurement in turbulent flow [84]....................67
Figure 3-2 Computational domain with boundary conditions..................................76
Figure 3-3 Axial velocity distribution of supersonic oxygen jet at 772K ambient
temperature for coarse, medium, fine and very fine grid level. ..................................80
Figure 3-4 Velocity distribution on centre axis with $k - \varepsilon$ turbulence model including
compressibility correction [17]....................................................................................81
Figure 3-5 Velocity distribution on centre axis with temperature corrected $k - \varepsilon$
turbulence model of Abdol-Hamid et al.[88]...............................................................82
Figure 3-6 Velocity distribution on centre axis with the proposed modified $k - \varepsilon$ model.
.......................................................................................................................................83
Figure 3-7 Two dimensional velocity distribution at different ambient temperatures. ..83
Figure 3-8 Dynamic pressure distribution on the centre axis of the jet with the proposed
model..............................................................................................................................84
Figure 3-9 Radial distribution of supersonic jet at different ambient temperatures at
$xde = 5$..........................................................................................................................86

xiv
Figure 3-10 Radial distribution of supersonic jet at different ambient temperatures at $x_{de} = 22.5$................................................................................................................................. 86
Figure 3-11 Radial distribution of supersonic jet at different ambient temperatures at $x_{de} = 50$................................................................................................................................................. 87
Figure 3-12 Spreading rate of supersonic jet at different ambient temperatures. ........ 88
Figure 3-13 Temperature distribution on centre axis with the proposed model......... 89
Figure 3-14 Values of coefficient $C\mu$ at 285K ambient temperature using present model. .................................................................................................................................................. 90
Figure 3-15 Values of coefficient $C\mu$ at 772K ambient temperature using present model. .................................................................................................................................................. 90
Figure 3-16 Values of coefficient $C\mu$ at 1002K ambient temperature using present model........................................................................................................................................... 91
Figure 3-17 Values of coefficient $C\mu$ at 1800K ambient temperature using present model........................................................................................................................................... 91
Figure 3-18 Values of coefficient $C\mu$ at 285 K ambient temperature using Heinz model [17]....................................................................................................................................................... 92
Figure 3-19 Values of coefficient $C\mu$ at 772 K ambient temperature using Heinz model [17]....................................................................................................................................................... 92
Figure 3-20 Values of coefficient $C\mu$ at 1002K ambient temperature using Heinz model [17]....................................................................................................................................................... 92
Figure 3-21 Comparison of velocity ratio with equation (3.23) as a function of $papgx_{de}$. ...................................................................................................................................................... 94
Figure 3-22 Coherent length of supersonic jet at different ambient temperatures. ...... 95
Figure 3-23 Variation of Blowing number with nozzle bath distance at different ambient temperatures. ................................................................................................................................................. 97
Figure 3-24 Computational domain with boundary conditions.......................... 103
Figure 3-25 Predicted axial velocity distributions at the jet centreline of shrouded oxygen jet using coarse, medium and fine grid levels................................. 107
Figure 3-26 Axial velocity distributions at the jet centreline of supersonic oxygen jet with and without shrouding flame.......................................................... 108
Figure 3-27 Predicted density distribution in case of shrouded oxygen jet........... 109
Figure 3-28 Axial velocity distributions at the jet centreline of supersonic nitrogen jet with and without shrouding flame......................................................... 112
Figure 3-29 Half jet width of the supersonic oxygen and nitrogen jets with and without shrouding flame................................................................. 112
Figure 3-30 Predicted axial static temperature distributions at the jet centreline of shrouded oxygen and nitrogen jet......................................................... 114
Figure 3-31 Shape of the combustion flame for shrouded oxygen jet................. 115
Figure 3-32 Shape of the combustion flame for shrouded nitrogen jet............... 115
Figure 3-33 CFD plot of vorticity contour for conventional oxygen jet........... 117
Figure 3-34 CFD plot of vorticity contour for shrouded oxygen jet.................. 118
Figure 3-35 Radial distribution of vorticity magnitude at different axial locations for both conventional and shrouded oxygen jet........................................... 118
Figure 3-36 CFD plot of turbulent shear stress for conventional oxygen jet......... 119
Figure 3-37 CFD plot of turbulent shear stress for shrouded oxygen jet............ 119
Figure 3-38 Predicted axial mass fraction distributions at the jet centreline of conventional and shrouded oxygen jet................................................................. 120
Figure 3-39 Predicted radial distributions of the CO₂ mass fractions at different axial locations for shrouded oxygen jet......................................................... 121
Figure 3-40 Predicted radial distributions of the CO\textsubscript{2} mass fractions at different axial locations for shrouded nitrogen jet. ................................................................. 122
Figure 3-41 CFD plot of the CO\textsubscript{2} mass fractions for shrouded oxygen jet .......... 122
Figure 4-1 Computational domain with boundary conditions. ................................. 135
Figure 4-2 Axial velocity distributions of supersonic jet at two different shrouding gas flow rates. ............................................................................................................................... 140
Figure 4-3 Axial dynamic pressure distribution of supersonic jets at two different shrouding gas flow rates. ............................................................................................................................... 141
Figure 4-4 Calculated depth of penetration for different grid levels. ......................... 144
Figure 4-5 Mechanism of droplets generation. ......................................................... 147
Figure 4-6 CFD prediction of the velocity distribution resulting from top jetting ...... 149
Figure 4-7 Variation of \(\eta\) value with lance heights. ........................................... 149
Figure 4-8 Variation of droplet generation rate with lance heights ....................... 151
Figure 4-9(a) Volume fraction and (b) velocity vectors at time \(t=3.9s\) for higher shrouding flow case. .......................................................................................................................... 153
Figure 4-10(a) Volume fraction and (b) velocity vectors at time \(t=3.9s\) for lower shrouding flow case. .......................................................................................................................... 153
Figure 4-11 Variation of depth of penetration with liquid density. ......................... 154
Figure 5-1 The two dimensional thin slice model for the experimental study .......... 158
Figure 5-2 Grid on the furnace wall to measure the depth of the cavity ................. 159
Figure 5-3 Schematic of experimental Set-up. .......................................................... 160
Figure 5-4 Splash collector used in the study. ....................................................... 160
Figure 5-5 Depth of penetration vs flow rate at 35 and 45 degree lance angles and 0.105m lance height. Data points are an average of 30 penetration depth measurements,
the error bars are the standard deviation for the 30 measurements and were found to be approximately 10% of the average penetration depth. .................................................. 164

Figure 5-6 Splashing rate vs flow rate at (a) 65 mm (b) 105 mm and (c) 165 mm lance heights. Data points are an average of 6 splashing rate measurements; the error bars are the standard deviation for the 6 measurements and were found to be approximately 9% of the average splashing rate. .......................................................... 165

Figure 5-7 Cavity formed at 65mm lance height, 0.71Nm³/min flow rate and at a lance angle of (a) 35 degree and (b) 45 degree.......................... 167

Figure 5-8 Splashing vs lance angle at 0.71Nm³/min flow rate. Data points are an average of 6 splashing rate measurements; the error bars are the standard deviation for the 6 measurements and were found to be approximately 9% of the average splashing rate.......................... 168

Figure 5-9 Splashing rates as a function of lance heights.................................................. 169

Figure 5-10 Wave progression in angled jetting from CFD and experimental study… 171

Figure 5-11 Splash distribution on the wall at (a) 65mm and (b) 165mm lance heights. .......................................................... 173

Figure 5-12 Schematic of inclined jetting .......................................................... 174

Figure 5-13 Dimensionless jet momentum vs cavity depth............................................ 176

Figure 5-14 Variation of critical depth of penetration with angle of inclination. ...... 177

Figure 5-15 Variation of η with lance angle.................................................. 179

Figure 5-16 Variation of Blowing number with lance angle........................................ 182

Figure 5-17 Wall splashing rate at different blowing number at 65 mm and lance height. .......................................................... 183

Figure 5-18 Wall splashing rate at different blowing number at 165 mm lance height. .......................................................... 183
Figure 6-1 Computational domain with boundary conditions.................................. 190
Figure 6-2 Variation of depth of penetration with flow rates at (a) 35 degree and (b) 45 degree lance angle................................................................. 193
Figure 6-3 Variation of diameter of depression with flow rates at (a) 35 degree and (b) 45 degree ................................................................. 195
Figure 6-4 Calculated depth of penetration at different grid levels......................... 196
Figure 6-5 Wave progression in angled jetting from CFD and experimental study (chapter 5)................................................................. 197
Figure 6-6 Predicted velocity distribution of inclined jetting. ................................. 199
Figure 6-7 Variation of $\eta$ value with lance angle ............................................. 200
List of Tables

Table 2-1 BOF Steelmaking event times [2] ............................................................... 12
Table 2-2 Tap-to-tap cycle of EAF steelmaking process [2]. ....................................... 18
Table 3-1 Different types of turbulence model [84] .................................................. 69
Table 3-2 Coefficients and constants of equation (3.16) ............................................. 73
Table 3-3 Boundary conditions for the modelling of free supersonic jet. .................... 77
Table 3-4 Under-relaxation factors ....................................................................... 79
Table 3-5 Boundary conditions for coherent oxygen jet simulation. ....................... 104
Table 3-6 Boundary conditions for coherent nitrogen jet simulation. ....................... 105
Table 3-7 List of user defined subroutines used in the simulation. ......................... 106
Table 4-1 Boundary conditions for modelling of top impinging jet. ....................... 138
Table 4-2 Physical properties of the fluids ............................................................. 138
Table 4-3 Under-relaxation factors for gas impingement study ............................... 139
Table 4-4 Average penetration depth for different shrouding flow rates. ............... 142
Table 4-5 CFD prediction of the turbulent free jet ($U_j$) and critical gas velocity ($U_g$) at different lance heights .................................................. 150
Table 4-6 Cavity dimensions at different lance heights ......................................... 152
Table 5-1 Dimensional analysis of the oxygen lance [53]. ..................................... 157
Table 5-2 Operating conditions selected for the present study. .............................. 162
Table 5-3 Critical impact velocities at different lance angles .................................. 181
Table 5-4 Dimensional analysis of the full-scale furnace ....................................... 184
Table 6-1 Lance height, lance angle and the nozzle exit velocity in the present simulation .......................................................... 191
Table 6-2 Physical properties of fluids at 303K temperature ................................. 192
Nomenclature

\[
A = \text{Cross-sectional Area (m}^2) \\
\dot{a} = \text{speed of the sound (m/s)} \\
c_p = \text{Specific heat at constant pressure (Jkg}^{-1}K^{-1}) \\
d_e = \text{Nozzle exit diameter (m)} \\
d_t = \text{Nozzle throat diameter (m)} \\
D_i = \text{Diffusion coefficient of species "i" (m}^2\text{s}^{-1}) \\
E = \text{Heat transferred by radiation (W)} \\
e = \text{Specific enthalpy per unit mass (J/kg)} \\
F = \text{Force (N)} \\
F_g = \text{Volumetric gas flow rates (Nm}^3\text{s}^{-1}) \\
m = \text{mass (kg)} \\
Fr_m = \text{Modified froude number (\text{--})} \\
g = \text{Gravitational acceleration (m/s}^2) \\
h = \text{lance height (m)} \\
h' = \text{axial distance between nozzle exit plane and undisturbed surface in inclined jet (m)} \\
H = \text{Total enthalpy per unit mass (J/kg)} \\
K = \text{Thermal conductivity (Wm}^{-1}K^{-1}) \\
\ell = \text{turbulence length scale (m)} \\
L_1 = \text{Height of the bottom liquid layer (m)} \\
L_2 = \text{Height of top liquid layer (m)} \\
M = \text{Momentum (N)} \\
Ma = \text{Mach number (\text{--})} \\
M_g = \text{Gradient Mach number (\text{--})}
\]
$M_t = \text{Turbulence Mach number} (-)$

$\dot{m} = \text{mass flow rate (kg/s)}$

$N_{we} = \text{Nominal weber number} (-)$

$N_B = \text{Nominal Blowing number} (-)$

$P = \text{Pressure of fluid (Pa)}$

$P_d = \text{Dynamic pressure of the jet (Pa)}$

$Pr_t = \text{Turbulent Prandtl number} (-)$

$p = \text{depth of penetration (m)}$

$p_c = \text{critical depth of penetration (m)}$

$q_i = \text{conduction heat transfer (Wm}^{-2}\text{)}$

$R = \text{Universal gas constant (Jkg}^{-1}\text{K}^{-1}\text{)}$

$R_B = \text{Droplet generation rates (kg/s)}$

$r = \text{distance in radial direction (m)}$

$r_{1/2} = \text{half jet width (m)}$

$S = \text{Mean shear rate (s}^{-1}\text{)}$

$S_p = \text{Spreading rate} (-)$

$Sc_t = \text{Turbulent schmidt number} (-)$

$T = \text{Temperature (K)}$

$T_e = \text{Temperature at the nozzle exit (K)}$

$T_a = \text{Ambient temperature (K)}$

$T_t = \text{Total temperature (K)}$

$t' = \text{Fluctuating component of temperature (K)}$

$t = \text{time (s)}$

$U = \text{Velocity (m/s)}$

$U_0 = \text{Nozzle exit velocity (m/s)}$

$U_j = \text{Turbulent jet axial velocity (m/s)}$
\( U_g \) = critical tangential velocity (m/s)
\( u_i, u_j \) = Turbulent velocity (m/s)
\( x \) = distance in x direction (m)
\( Y_i \) = Mass fraction of species i (–)

**Greek letters**

\( \zeta \) = Vorticity (s\(^{-1}\))
\( \alpha \) = Volume fraction(–)
\( \rho \) = density (kg/m\(^3\))
\( \rho_g \) = gas density (kg/m\(^3\))
\( \rho_l \) = liquid density (kg/m\(^3\))
\( \rho_a \) = ambient fluid density (kg/m\(^3\))
\( \rho_{L_1} \) = Density of the bottom liquid layer (kg/m\(^3\))
\( \rho_{L_2} \) = Density of the top liquid layer (kg/m\(^3\))
\( \sigma \) = surface tension (N/m)
\( \sigma_s \) = Stefan – Boltzmann constant (Wm\(^2\)K\(^{-4}\))
\( \mu_l \) = viscosity of liquid (N – s/m\(^2\))
\( \mu_t \) = Turbulent viscosity (N – s/m\(^2\))
\( k \) = Turbulent kinetic energy (m\(^2\)/s\(^2\))
\( \varepsilon \) = turbulence dissipation rate (m\(^2\)/s\(^3\))
\( \Gamma \) = Diffusion coefficient(m\(^2\)s\(^{-1}\))
\( \tau_{ij} \) = Viscous stress(N/m\(^2\))
\( \theta \) = lance inclination angle (degree)
\( \gamma_{L_1} \) = Specific weight of bottom liquid layer (kg/m\(^2\)s\(^2\))
\( \gamma_{L_2} \) = Specific weight of top liquid layer (kg/m\(^2\)s\(^2\))
\( \delta_{ij} \) = Kronecker delta (–)
Chapter 1

1. Introduction
1.1 Background

Steel is used in almost every area of modern age such as in construction of roads, buildings, infrastructures, railways, vehicles, ships, machineries and tools. Because of the wide range of applications and critical role played by steel in infrastructural and overall economic development, the steel industry is often considered to be an indicator of economic progress. Figure 1-1 shows that annual steel production in the world increased steadily and linearly from the year 1970 to 2000. After that there was a sharp increase in the annual steel production rate which was largely due to the economic growth in China.

![Annual steel productions](image)

Figure 1-1 The world annual steel productions from 1970 to 2010 [1].

Though steel had been produced by various methods before nineteenth century, the modern era of steelmaking began with the introduction of Bessemer process in 1858 which removes impurities from the molten iron by oxidation with air blown through the
molten iron [2]. At present, steel is produced from two different routes: Basic oxygen furnace (BOF) steelmaking and electric arc furnace (EAF) steelmaking. The first commercial operation of BOF steelmaking was in the early 1950’s at Linz and Donawitz [3]. This manner of steelmaking is also known as Linz-Donawitz or LD process named after the places. In this process, iron ores are melted in the blast furnace to produce liquid pig iron and then poured into the steelmaking converter. After that pure oxygen is blown through the molten iron bath, instead of air as in Bessemer process, in order to reduce the carbon content and other impurities [3].

Over the past 30 years the use of the EAF route for the production of steel has grown considerably. In this process, scrap materials are used to produce steel which is a key element in achieving the goals of sustainable development [4]. Mainly electrical energy along with some chemical energy is used to melt the scrap materials and then oxygen jet is used to purify the molten metal and produce high quality steel. In developed countries like the United States, Europe and Japan, the amount of obsolete scrap in relationship to the amount of steel required increased, reducing the price of scrap relative to that of hot metal produced from ore and coal. This economic opportunity arising from low cost scrap and the lower capital cost of an EAF compared to integrated steel production (BOF) lead to the growth of the mini-mill or scrap based EAF producer [2].

Both in BOF and EAF steelmaking oxygen is injected under supersonic conditions into the furnace in order to reduce the level of carbon in the metal bath and to oxidize impurities like phosphorus, silica and other elements to slag [3]. Supersonic gas jets are used to produce higher depth of penetration on the liquid metal surface and obtain better gas-liquid mixing inside the furnace which may result in higher reaction rates. In addition, supersonic gas jets generate metal droplets upon impingement on the liquid bath which have both positive and negative effects on the metal processing operations. The available interfacial area of the generated droplets is very large which in turn enhance the rates of heat transfer and chemical reactions. If the velocity of the supersonic gas jet is very high, the generated metal droplets splashes onto water-cooled panels, slag/steel build-up on the lances which is detrimental to water-cooled panels, refractories and lance tips [5] and results in the loss of bulk metal. Hence, knowledge of supersonic jet behaviour and the jet-liquid interactions are required for the optimization of the process and make it more efficient.
The environment inside the furnace is very harsh making it difficult to make direct observations and measurements. As a result, a number of cold modelling and mathematical modelling studies have been carried out in last 50 years to understand the complex fluid flow phenomenon inside the steelmaking furnace. Almost all of them used incompressible air jet and water to understand the fluid flow behaviour inside the furnace. But the behaviour of the supersonic jet at steelmaking temperature is different from the incompressible air jet of cold modelling studies. Knowledge of jet velocity and oxygen mass fractions at the liquid surface is important for predicting the cavity depth and chemical reactions at the impingement point for both the BOF and EAF steelmaking processes. Some numerical studies of the jet at steelmaking temperature are available in the literature but those were not validated against the experimental data. Hence, a properly validated numerical model is required to predict jet properties at steelmaking conditions.

In the late 1990’s, coherent supersonic jet was introduced in EAF steelmaking process replacing the conventional supersonic jet. Coherent jet is formed by shrouding the central supersonic jet with a combustion flame which is created by fuel and oxidant. Because of the flame envelope, the entrainment of the surrounding gas into the supersonic jet is reduced, leading to a higher potential core length of the supersonic jet. The longer potential core length makes it possible to install the nozzle far from the liquid surface and increase the nozzle tip life. It is also claimed that coherent jet produces less splashing on the furnace wall compared with conventional supersonic jets. Although the steelmaking industries have been using the coherent jet since late 1990’s, limited research have been performed to understand the physics of supersonic coherent jet characteristics and how it can contribute to minimize the wall splashing which is very important in order for further improvement of this technology.

In an attempt to quantify the rate of droplet generation from the jet-liquid interactions, couple of dimensionless numbers [6, 7] have been proposed by the previous researchers. Among those, the Blowing number \( (N_B) \) theory is superior in physical sense because it estimates how many times the Kelvin-Helmholtz interface instability criterion has been exceeded. According to this theory, the higher the Blowing number, the more is the droplet generation rate. However, the available literature shows that the droplet generation rate also increases at higher lance inclination angles [8]. The existing
Blowing number theory does not include the effect of lance inclination angle. Moreover, at very low lance height the Blowing number on the liquid surface is higher because of higher jet impingement velocity which in turn predicts higher droplet generation rate. But it was reported [6, 9] that droplet generation rate decreases at very low lance height. Hence, a proper modification of Blowing number theory can contribute to more accurate prediction of the droplet generation rate at those conditions.

1.2 Aim and Scope of This Study

The aim of this research is to contribute to the understanding of the supersonic gas jet behaviour, jet liquid interaction and droplet generation using the computational fluid dynamics (CFD) and physical modelling. In order to achieve this, the following actions are undertaken step by step:

- A CFD investigation of the effect of higher ambient temperatures on supersonic jet characteristics.
- A CFD investigation of the effect of shrouding combustion flame on coherent supersonic jet characteristics.
- A CFD investigation of the shrouded jet-liquid interaction in case of top jetting (BOF).
- An experimental and subsequent CFD investigation of jet-liquid interactions and splashing in case of inclined jetting.

All CFD models have been developed using commercial CFD software AVL FIRE v2008 and AVL FIRE v2009. The limitations of the CFD model in predicting jet properties, jet-liquid interactions and droplet generation rate are discussed and the necessary modifications are proposed. Several assumptions will be made for the CFD modelling and the validity of these assumptions is argued in terms of available experimental data.
The scope of this research is limited to the experimental data available. All the CFD models are validated against the available experimental data and then efforts have been made to predict the flow behaviour at steelmaking temperature wherever possible.

1.3 Thesis Overview

A literature review on BOF and EAF steelmaking has been presented in Chapter 2 to give a general background about the process. It is then followed by the theory of the supersonic gas jet and the structure of coherent supersonic jet. Lastly, the chapter explores available current literature on the experimental and numerical modelling of supersonic jet, liquid flow, liquid surface deformation and droplet generation, resulting from the jet impingement.

Chapter 3 starts with the description of the governing equations required for solving the supersonic gas jets without shrouding flow at higher ambient temperatures. Then the turbulence modelling of the jet at higher ambient temperature has been discussed. It is then followed by the detailed description of boundary conditions, computational procedure and finally the results and discussions. In the second half of Chapter 3, modelling of the coherent supersonic jet has been discussed. The description includes modification of energy equation for coherent jet, details on combustion modelling procedure, necessary boundary conditions, computational procedure and finally the results and discussions.

Chapter 4 consists of the CFD simulation of shrouded supersonic jet impingement on a liquid surface. The effect of shrouding gas flow rate on the depth of penetration has been investigated and validated against the experimental data. The droplet generation mechanism has been investigated in detail and compared qualitatively with the established theory.

Chapter 5 describes the experimental investigation of droplet generation rate at different operating conditions (lance height, lance angle and flow rates). The critical penetration depth for the onset of splashing at different lance angle has been investigated. A modification of the Blowing number theory by incorporating the lance inclination angle
has been proposed. In Chapter 6, the numerical modelling of inclined jetting on liquid surface is presented.

Finally, the conclusions, drawn from the present study, are presented in Chapter 7 which is then followed by some suggestions for the future study on this project.

1.4 Publications From This Research

1.4.1 Journals


1.4.2 Conferences


Chapter 2

2. Literature Review
2.1 Overview of Steelmaking Processes

The entire process of steelmaking can be divided into two sequential steps: primary steelmaking in the furnaces and secondary steelmaking in a ladle [3]. In primary steelmaking, liquid iron that comes from the blast furnace is purified by blowing oxygen into the converter. This process is known as Oxygen steelmaking. In another approach, the recycled iron and steel scraps are melted using electric arcs inside the furnace and then purified by blowing oxygen which is known as Electric Arc Furnace (EAF) steelmaking process [3]. Both the BOF and EAF steelmaking process are described in brief in the following sections.

2.1.1 The Oxygen Steelmaking Process

In oxygen steelmaking process, high purity oxygen is used to refine a charge of molten iron and ambient scrap into steel of desired carbon content and temperature by oxidizing the dissolved impurities like carbon, silicon, manganese, phosphorus and sulphur [2]. The molten iron comes from the blast furnace where iron ore, coke and lime are used as raw materials. The molten iron usually contains 4-4.5wt% carbon and the previously mentioned impurities which make the metal too brittle for most engineering applications [10]. The main functions of the oxygen steelmaking process are to reduce the carbon to less than 0.1%, to reduce or control the sulphur and phosphorus, and finally, optimize the temperature of the liquid steel so that any further treatments prior to casting can be performed with minimal reheating or cooling of the steel [2, 10]. The exothermic oxidation reactions that occur during oxygen steelmaking generate a lot of heat energy which is removed by the addition of scrap materials and iron ores of calculated amount to obtain the desired final temperature [10].

2.1.1.1 Types of Oxygen Steelmaking Process

There are basically three variations of oxygen steelmaking process depending on the method of introducing oxygen gas into the liquid bath [2]:
- Top blowing process
- Bottom blowing process
- Combined blowing process

The most common and widely used configuration is the top blown converter which is also known as basic oxygen furnace (BOF) process. In this process, oxygen is blown via a water cooled lance, inserted from the top of the vessel, as shown in Figure 2-1(a). The blowing end of the lance is fitted with de Laval nozzle in order to deliver oxygen at supersonic velocity. The use of supersonic oxygen jet to refine hot metal was first introduced at Linz and Donawitz of Austria in 1949. That is why this process is also known as LD process, named after the two places [3]. In the bottom-blown converters, oxygen is introduced through the bottom of the vessel with the help of tuyeres. The

![Figure 2-1](image-url)
bottom-blowing process is also called the OBM, Q-BOP process depending upon the type of tuyere design. Each tuyere consists of two concentric pipes with the oxygen passing through the centre pipe and a coolant hydrocarbon passing through the annulus between the pipes as shown in Figure 2-1(b). Usually, natural gas or oil is used as coolant which chemically decomposes when introduced at high temperatures generated by oxidation reactions and hence, protects the tuyere tip from overheating [2]. One of their advantages is a very efficient metal-slag stirring throughout the oxygen blow [10]. In the combined blowing process, gases are blown both from the top lance and bottom tuyeres. Combined blowing process can be of different types such as K-BOP, LD-OTB, LD-STB, LBE etc. depending upon the amount of gas injected from the top and bottom and also on the type of tuyere and the gas injected [3]. Figure 2-1(c) shows three different types of bottom blowing converters: top lance with permeable elements, top lance with cooled bottom tuyeres and top lance with uncooled bottom.

### 2.1.1.2 Sequence of Operations Of BOF Steelmaking

In BOF steelmaking, steel is made in discrete batches called heats. Time required to produce a typical 250 ton heat varies in the range of 30-65 minutes [2]. The major event times in each heat is summarized below in Table 2-1:

Table 2-1 BOF Steelmaking event times [2].

<table>
<thead>
<tr>
<th>Events</th>
<th>Minutes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Charging scrap and hot metal</td>
<td>5-10</td>
</tr>
<tr>
<td>Refining-blowing oxygen</td>
<td>14-23</td>
</tr>
<tr>
<td>Sampling-chemical testing</td>
<td>4-15</td>
</tr>
<tr>
<td>Tapping</td>
<td>4-8</td>
</tr>
<tr>
<td>Pouring slag off at furnace</td>
<td>3-9</td>
</tr>
</tbody>
</table>
The sequence of operations for BOF steelmaking is shown in Figure 2-2:

Figure 2-2 Schematics of operational steps in BOF steelmaking process [2].

At first the furnace is tilted by about 30-40° angle from the vertical position and the scrap is charged into the vessel with the help of a charging bucket. Scrap basically consists of recycled iron or steel. Hot metal is then poured on the scrap [3]. In some cases hot metal is desulphurized in separate vessel before charging into the BOF steelmaking converter. Because the oxidizing condition inside the furnace is not suitable for sulphur removal. The sulphur level in desulphurized hot metal can be as low as 0.002wt% [2]. The pouring operation, which generates considerable dust, is accomplished under an enclosed hood equipped with an evacuation system and a baghouse [2].

After charging of scrap and hot metal, the vessel is then tilted back to its normal upright position and water cooled oxygen lance is gradually lowered up to a specified distance from the liquid metal. Blowing is started simultaneously while lowering the lance [3]. The oxygen is blown at supersonic velocities and the blow rate ranges from 560 to 1000Nm³/min [2]. Impurities such as carbon, silicon, manganese, phosphorus, sulphur, dissolved in the hot metal, are removed by oxidation. The energy required to raise the
scrap and hot metal temperature from 1250-1450°C to steelmaking temperatures of about 1600-1680°C is provided by exothermic oxidation reactions [3]. The major oxide-forming reactions are as follows [10]:

\[
\begin{align*}
2[P] + \frac{5}{2}(O_2)_g &= (P_2O_5) \quad 2-1 \\
[Fe] + \frac{1}{2}(O_2)_g &= (FeO) \quad 2-2 \\
[Mn] + \frac{1}{2}(O_2)_g &= (MnO) \quad 2-3 \\
[Si] + (O_2)_g &= (SiO_2) \quad 2-4 \\
[C] + \frac{1}{2}(O_2)_g &= (CO) \quad 2-5 \\
2[Fe] + \frac{3}{2}(O_2)_g &= (Fe_2O_3) \quad 2-6
\end{align*}
\]

A typical change in melt composition during the blow is shown in Figure 2-3. The silicon dissolved in the metal is almost completely oxidized in the first 3-4 minutes of the blow. In contrast to silicon, the transfer of sulphur, phosphorus, manganese and carbon takes place over the entire period of blow [3].

Figure 2-3 Change in melt composition during the blow [3].
Soon after the oxygen is turned on, flux additions are also started. In oxygen steelmaking, normally CaO (from burnt lime) and MgO (from dolomitic lime) are used as flux. The CaO is used to dissolve the metal oxides (SiO$_2$, MnO, FeO, Fe$_2$O$_3$) and form a basic liquid slag that is able to remove sulphur and phosphorus from the metal. The dolomite is used to saturate the slag with MgO and reduce the dissolution of the dolomite furnace refractories. In some case fluorspar is used to reduce slag viscosity [2].

The position of the lance is important for proper functioning of the process. If the lance is too high, the slag will be over stirred and over-oxidized with higher FeO percentages. This will cause higher than normal yield losses and lower tap alloy efficiencies due to oxidation losses. Further, the rate of carbon removal is reduced and becomes erratic. When the lance is too low, carbon removal increases somewhat, slag formation, slag reactivity, and FeO are reduced and sulphur and phosphorus removal problems often occur. If the lance is very low, then spitting of metal droplets or sparking occur which cause severe and dangerous metallic deposits, called skulls, on the lance and the lower waste gas hood. Obviously, there is an optimum lance height which varies from shop to shop depending on the process conditions [2].

At the end of the blow the lance is raised and the furnace is then rotated towards the charging side nearly 90° to do chemical analysis and temperature measurement. Based on the chemical laboratory results, a decision is taken whether the heat is ready to tap or requires corrective action. If a corrective action is required, the furnace is set upright. Oxygen is re-blown into the furnace, with or without coolants or fluxes, to arrive at the desired (aim) chemistry and temperature [2]. The typical end point composition of steel is 0.04-0.06wt% carbon, 0.2wt% manganese, 0.02wt% phosphorus and 0.015wt% sulphur. The manganese, sulphur and phosphorus contents are function of input composition of scrap and hot metal [3]. Once the heat meets the temperature and chemistry requirements, the furnace is rotated towards the taphole side and the refined steel is tapped into a ladle. Near the end of tapping a vortex may develop near the draining taphole and entrain some of the slag into the ladle. Furnace slag contains high FeO, which reduces desulphurization in the ladle. Also, the P$_2$O$_5$ present in BOF slag is a source of phosphorus carried into the ladle. So slag carryover from the BOF into the ladle must be minimized [2]. After tapping, the slag is either taken out by tilting the converter or splashed on the wall of the furnace to coat the lining and thereby extend its
life. This slag splashing (coating) maintenance is typically performed by blowing nitrogen through the oxygen lance for two to three minutes [2].

2.1.2 Electric Arc Furnace Steelmaking Process

In EAF steelmaking, high powered electric arc is used to melt the solid scrap materials inside the furnace instead of blast furnace to melt iron ore as in oxygen steelmaking process. Although, the latter one still dominates the world production of steel, the EAF route accounts for a larger and larger portion every year as shown in Figure 2-4. This is due to the advances in EAF technology which has reduced the tap to tap time and electrical energy consumption. In 2001-2003, approximately one-third of the annual worldwide steel production was made via the EAF route and then it dropped down recently due to the economic recession. The production capacity of an EAF usually varies between 50 and 150 tons at different plants [10] but it can be made of almost any
size from 1 ton to 400 tons whereas an oxygen steelmaking converter less than 30 tons are not economical [3]. Normally, recycled steel scrap is used as raw materials saving the consumption of virgin raw materials and energy, which is beneficial from both economic and environmental point of view. On an average, 40MWh electrical energy is required to melt 100 tons of recycled steel scrap [2].

Figure 2-5 shows a schematic of electric arc furnace. An electric arc furnace consists of a furnace shell with a roof on top. The furnace shell is lined with ceramic bricks (usually carbon bonded magnesia bricks) insulating the furnace from the liquid steel. On the upper side of the wall and on the inside of the roof, water cooled elements are used instead of ceramic insulation. Because of the positions of these water cooled panels, there is no direct contact with the liquid steel [10].

![Schematic of an electric arc furnace](image)

The EAF operating cycle is known as the tap-to-tap cycle which is made up of the following operations: furnace charging, melting, refining, de-slagging, tapping and furnace turnaround. Modern operations aim for a tap-to-tap cycle of less than 60 minutes. A typical 60 min tap to tap cycle is shown in Table 2-2:
Table 2-2 Tap-to-tap cycle of EAF steelmaking process [2].

<table>
<thead>
<tr>
<th>Steps</th>
<th>Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>first charge</td>
<td>3 minutes</td>
</tr>
<tr>
<td>first meltdown</td>
<td>20 minutes</td>
</tr>
<tr>
<td>second charge</td>
<td>3 minutes</td>
</tr>
<tr>
<td>second meltdown</td>
<td>14 minutes</td>
</tr>
<tr>
<td>Refining</td>
<td>10 minutes</td>
</tr>
<tr>
<td>Tapping</td>
<td>3 minutes</td>
</tr>
<tr>
<td>Turnaround</td>
<td>7 minutes</td>
</tr>
<tr>
<td>Total</td>
<td>60 minutes</td>
</tr>
</tbody>
</table>

At the beginning of the cycle, the furnace roof and the electrodes are raised and are swung out to the side of the furnace. Then the furnace is charged with recycled steel scrap by using scrap bucket. The scraps are layered in the bucket according to size and density in order to ensure rapid formation of a liquid pool in the hearth. Other considerations include minimization of scrap cave-ins which can break electrodes and ensuring that large heavy pieces of scrap do not lie directly in front of burner ports which may result in blow-back of the flame onto the water-cooled panels [2]. The charge can include lime and carbon or these can be injected into the furnace during the heat. After charging, the roof and the electrodes are swung back into place over furnace. The roof is then closed and the electrodes are lowered towards the scrap. Then the electrical power is switched on and electrical energy is transformed into heat as arcing takes place between the electrodes and the solid scrap. This is the commencement of melting period. At the start of melting the arc is erratic and unstable. Wide swings in current are observed accompanied by rapid movement of the electrodes. As the furnace atmosphere heats up, the arcing tends to stabilize. Once the molten pool is formed, the arc becomes quite stable and the average power input increases.
In modern electric arc furnace chemical energy is also supplied to the furnace for melting operation along with the electrical energy but the later one is the major contributor. Approximately 60% of the total energy input in EAF is supplied by electricity [2]. Chemical energy is supplied via oxy-fuel burners which burn natural gas using oxygen. Once a molten pool is formed the power is switched off, the furnace roof is opened again, and second bucket of scrap is charged. Then the electrical switch is turned on again and melting of scrap starts. When all the scraps are melted, heating continues for some time to superheat the molten steel to the target tapping temperature [10]. Then supersonic oxygen jet is lanced directly into the bath which is the commencement of refining period. Unlike BOF, in EAF oxygen is injected at a certain angle from the vertical through the sidewall fixed injectors because in EAF electrodes are inserted from the top of the furnace. Most impurities such as, aluminium, silicon, manganese, phosphorus sulphur and carbon are removed by oxidation reactions which are described in previous section. The metallic oxides which are formed during the oxidation reactions transfer to the slag above the liquid steel. At the end of refining period, bath temperature is measured and a bath sample is taken for chemical analysis. If the correct composition is reached, the furnace is tilted and de-slagging operations are carried out to remove the slag through the slag door [2]. Then the furnace is tilted on the other side of the slag door, taphole is opened and molten steel is poured into a ladle where the secondary steelmaking operations are carried out [2].

2.1.2.1 Foamy Slag Practice

The development of foamy slag practice is one of the key technologies that has advanced EAF steelmaking. When the charge is completely melted, the furnace sidewalls can be exposed to high radiation from arc. A foamy slag is created to cover the arc, protect the sidewall from radiation and transfer energy to the bath as shown schematically in Figure 2-6. Oxygen is injected with coal to foam up the slag by producing CO gas in the slag [2]. In some cases only carbon is injected and carbon reacts with FeO in the slag to produce CO gas. The reactions of CO formation are:
\[ [C] + [O] = (CO)_g \] 

\[ (FeO) + [C] = (CO)_g + [Fe] \]

Some of the benefits attributed to foamy slag are decreased heat losses to the sidewalls, improved heat transfer from the arcs to the steel that allows for higher rate of power input, reduced power and voltage fluctuations, reduced electrical and audible noise, increased arc length (up to 100%) without increasing heat loss and reduced electrode and refractory consumption [2].

![Conventional vs Foaming Slag](image)

Figure 2-6 Effect of slag foaming on arc radiation [2].

### 2.2 Theory of Supersonic Gas Jet

A gas jet is a stream of fluid forcefully shooting forth from a nozzle into an open environment. Depending on the velocity of the fluid stream, a gas jet can be divided into three different categories:

*Subsonic flow:* The velocity of the fluid stream is lower than the speed of sound in that particular condition.

*Sonic flow:* The velocity of the fluid stream is equal to the speed of sound in that particular condition.
Supersonic flow: The velocity of the fluid stream is higher than the speed of sound in that particular condition.

As mentioned earlier, supersonic gas jets are used to decarburize and purify the liquid iron during refining period in steelmaking process. De Laval nozzles are used to accelerate gas jets to supersonic velocities [3]. The de Laval nozzle is a convergent divergent nozzle and was named after the inventor Gustaf de Laval. Figure 2-7 shows the schematic of a de Laval nozzle.

A properly designed Laval Nozzle accelerates the gas jet to the desired supersonic velocity for steelmaking by converting the available pressure energy of the gas into kinetic energy [3] as shown in Figure 2-7. The mass flow rate through the nozzle can be written as,

\[ \dot{m} = \rho AU = \text{constant} \quad 2-9 \]

where \( \rho \) is the density \((kg/m^3)\) of gas, \( A \) is the cross-sectional area of the nozzle and \( U \) is the velocity \((m/s)\) of the gas. It is assumed that the density, pressure and velocity of the gas inside the nozzle changes only in the direction of the flow (i.e. one-dimensional flow). The mass flow rate of the gas is constant at each point along the flow inside the nozzle. Differentiating equation (2.9) and dividing the resultant by the mass flow rate we obtain [11],

\[ \frac{d\rho}{\rho} + \frac{dA}{A} + \frac{dU}{U} = 0 \quad 2-10 \]

Due to the high speed of the gas and short length of the nozzle, the frictional losses are assumed small, the flow is assumed adiabatic and the entropy of the gas remains constant [11].
Figure 2-7 Schematic of a Laval Nozzle with Pressure and Mach number profile[11].
Assuming the change in potential energy negligible, the energy balance relation becomes,

\[ e + \frac{u^2}{2} = \text{constant} \]  

Where \( e \) is the static enthalpy (\( J/kg \)) per unit mass of the fluid.

Differentiating the equation (2.11)

\[ de + UdU = 0 \]  

From thermodynamics\([11]\),

\[ Tds = de - \frac{dp}{\rho} \]  

\[ de = \frac{dp}{\rho} \quad \text{[entropy is constant, } ds = 0] \]  

Substituting equation (2.14) into (2.12),

\[ \frac{dp}{\rho} + UdU = 0 \]  

In an adiabatic process, the speed of the sound can be expressed as \([11]\),
\[ a = \frac{\sqrt{dp}}{\sqrt{d\rho}} \] 2-16

Combining equations (2.10), (2.15) and (2.16) gives,

\[ \frac{dA}{A} = \frac{du}{u} \left( \frac{u^2}{a^2} - 1 \right) \] 2-17

The ratio of the gas velocity to the speed of the sound at any point is known as the Mach number:

\[ Ma = \frac{u}{a} \] 2-18

Substituting equation (2.18) into equation (2.17),

\[ \frac{dA}{A} = \frac{du}{u} (Ma^2 - 1) \] 2-19

From equation (2.19) it can be concluded that:

- At subsonic flow \((Ma < 1)\), a decrease in area will lead to an increase in velocity.
- At supersonic flow \((Ma > 1)\), an increase in area will lead to an increase in velocity.
- At sonic flow \((Ma = 1)\), \(dA=0\), the rate of change of area at the throat is zero where sonic velocity can be obtained.

Hence, in order to accelerate the fluid to supersonic velocities, the fluid must pass through converging section of the nozzle until the flow becomes sonic at a certain minimum area (throat area) and then through the diverging portion of the nozzle. The result is a converging-diverging nozzle as shown in Figure 2-7. The Laval nozzle should be designed and operated in such a way that the static pressure at the exit of the nozzle
is equal to the ambient pressure. Otherwise shock waves will be generated at the exit of the nozzle [11] to equalize the pressure and part of the jet kinetic energy is lost in this process which in turn results in lower impact pressure on the liquid surface. However, in steelmaking the exit pressure should always be slightly greater than the ambient pressure so that the dust-laden gases from the ambient are not sucked inside the nozzle which may cause wearing of the nozzle surface or blocking of the nozzle hole [3].

When a supersonic jet exits from a Laval nozzle, it interacts with the surrounding still air to produce a region of turbulent mixing as shown in Figure 2-8. This process results in an increase in jet diameter and decrease in jet velocity with increasing distance from nozzle exit.

Figure 2-8 Regions of a supersonic jet exiting from a Laval nozzle.

A supersonic jet issuing from a Laval nozzle can be divided into three different regions [12]:

1) Potential core region-In this region the axial velocity of the gas is constant and is equal to the nozzle exit velocity. The length of the potential core region is proportional to the upstream pressure and ambient temperature. The length of the coherent region is also known as potential flow core length or coherent length.
2) **Transition region**- This region starts when the turbulent mixing layer reaches the axis of the flow and then gradually emerges to the fully developed regime.

3) **Fully developed flow**- In this region, the flow becomes fully turbulent. The jet begins to spread at higher rate compared to the potential core region.

In numerical modelling of the supersonic jet, the key challenge is the modelling of turbulence. From the experimental study, it was observed that when the velocity of the jet is very high, the mixing of the jet with its surroundings is suppressed and the growth rate of the turbulent mixing region is reduced [13]. It became known in late seventies that the standard \(k-\varepsilon\) model [14] gives a poor prediction of the axial velocity profiles of a high speed turbulent axisymmetric jets [15]. This occurs because the standard \(k-\varepsilon\) model lacks the ability to reproduce the observed reduction in growth rate of the turbulent mixing region at higher speed. Pope [15] reported that the standard \(k-\varepsilon\) turbulence model overestimates the spreading rate of the turbulent round jet by about 40%. However, some modifications of the \(k-\varepsilon\) model have been proposed [16, 17] in last two decades in order to take into account the effect of compressibility by reducing the growth rate of turbulent mixing region. Sarkar et al.[16] suggested the addition of an extra dissipation term, known as compressible dissipation, with the solenoidal dissipation in turbulent kinetic energy equation. It was shown that their proposed model can predict the dramatically reduced growth rate of compressible mixing layer in case of high speed jet. Heinze [17] proposed a modification of the turbulent production term in \(k-\varepsilon\) equation. In this model, the turbulent production level decreases with increasing compressibility of the jet which in turn suppress the growth rate of mixing layer. These modifications made it possible to obtain the velocity distribution of the supersonic free jet at room ambient temperature accurately.

The ambient environment, where the high speed jet discharges, also has significant effect on the turbulent mixing layer of the jet. From theoretical analysis, Chatterjee [18] reported that at higher ambient temperatures the entrainment of the surrounding ambient fluid into the jet is lower which in turn reduces the growth rate of turbulent mixing layer as well as jet spreading rate. As a result, the potential core length of the jet increases at higher ambient temperatures. Allemand et al.[12] reported from their numerical study that at 2000K ambient temperature, the potential core length of the supersonic jet becomes approximately 3 times than that at room ambient temperature. Numerical
simulations carried out by Tago and Higuchi [19] and Katanoda et al.[20] also showed an increase in potential core length of the jet at higher ambient temperatures. Sambasivam and Durst [21] showed from their numerical study that spreading rate of the jet is higher in denser medium which results in smaller potential core region. However, none of these above mentioned numerical simulations [12, 19-21] were validated against experimental data. To the authors knowledge, only Sumi et al.[22] experimentally studied the behaviour of the supersonic oxygen jet at three different ambient temperatures-285K, 772K and 1002K. The velocity attenuation of the jet was restrained and the potential core length was extended at higher ambient temperatures.

2.2.1 Coherent Supersonic Jet

From the preceding section it is known that when a supersonic gas jet discharges from a nozzle, it entrains fluids from surrounding environment which results in decaying of the jet axial velocity. Hence, during oxygen blowing, the higher the distance between liquid surface and the nozzle exit, the greater the entrainment of surrounding fluid which in turn decreases the impact velocity as well as depth of penetration on the liquid surface. As a result, the mixing of gas and liquid inside the furnace decreases which also reduces the reaction rates because of small gas-liquid interfacial area. Hence, it is desirable to
locate the nozzle close to the liquid metal surface. But the disadvantage of this is the sticking of slag/metal droplets on the lance tip which results in poor tip life. To overcome this problem, coherent jet (also known as shrouded jet) technology has been introduced to the Electric Arc Furnace (EAF) steelmaking industry in the late 1990’s. Coherent gas jets are produced by shrouding the conventional supersonic jet with a flame envelope as shown in Figure 2-9 [23-25]. The flame envelope is created using a fuel and oxidant. Generally CH$_4$ and O$_2$ are used as fuel and oxidant respectively. Figure 2-10 shows the cross-sectional and front views of a shrouded jet nozzle [25]:

![Coherent Nozzle Diagram](image)

**Figure 2-10 (a) Cross-sectional and (b) front view of a coherent nozzle [25].**

In a shrouded nozzle, the central converging-diverging nozzle is surrounded by small holes arranged in two concentric rings as shown in Figure 2-10. Supersonic oxygen flows through the central converging-diverging nozzle, the inner ring of holes supplies CH$_4$ and the outer ring of holes supplies shrouding subsonic oxygen [25]. Because of the flame envelope, the entrainment of the surrounding gas into the supersonic jet is reduced, leading to a higher potential core length of the supersonic jet [25, 26]. The longer potential core length of the coherent supersonic jet makes it possible to install the nozzle far from the liquid surface. Recently, Mahoney [27] showed form their experiment that the potential core length of the coherent jet increases with increasing fuel (CH$_4$) flow rate up to a certain limit. It was concluded that the potential core length
of the coherent jet becomes insensitive to fuel flow rate if the fuel flow rate exceeds 10% of the supersonic jet flow rate. The reactive ambient was also found to increase the potential core length of the jet [27]. The coherent jet is claimed to produce less splashing than the conventional supersonic jet [28], although no experimental study is available in the literature regarding the splashing phenomenon in case of coherent jets. Meidani et al. [29] carried out the only experimental study of shrouded supersonic jet impinging on liquid surface but only measured the effect of shrouding gas on the depth of penetration in case of top jetting. The effect of shrouding gas on the splashing rate was not reported. Moreover, in their study compressed air was used as shrouding gas in place of combustion flame.

In modern EAF’s, the shrouding oxygen and fuel are used as burner during the melting period and thus increase the efficiency of the process [30]. Some numerical studies [30-34] of supersonic jets with shrouding flame are available in the literature showing the axial velocity and Mach number distribution of coherent jet. Harris and Holmes [30] performed numerical simulation of a commercial coherent supersonic lance system known as KT injector [30]. Their numerical results showed that the shrouded combustion flame, created by natural gas and oxygen, extends the potential core region to 50 nozzle exit diameters far from the lance exit plane. It was also reported that the coherent length can be increased by minimizing the turbulence inside the jet. Candusso et al. [31] reported from their numerical simulation that the commercial coherent injector developed by MORE can maintain supersonic velocity up to 2 meters from the nozzle exit plane. In their study, natural gas was not used for shrouding the jet. The CO generated inside the furnace during post-combustion was claimed to act as a shrouding flame for the main supersonic jet. Lv et al. [34] carried out couple of numerical simulations of the coherent supersonic jet by changing the relative position of the shrouding oxygen and fuel gas injector holes. It was concluded that the change in relative position of the holes has no evident effect on the potential core length of the supersonic jet except when the nozzles were worn after multiple use.

Most of the numerical studies [30, 31, 34] were not validated against experimental data. The numerical simulation of coherent jet, carried out by Jeong et al. [32] was found to underpredict the potential core length of central supersonic oxygen jet when compared with the experimental study of Anderson et al. [25]. They attributed two dimensionality
of the numerical simulation for the underprediction of the coherent length. Liu et al. [33] performed both the experimental and numerical study to investigate the flow behaviour of the coherent jet. They proposed a modification for the model constant $C_1 = 1.45$ and $C_2 = 1.88$ of the standard $k - \varepsilon$ turbulence model. The numerical results with the modified turbulence model were found to predict the experimental results more accurately.

### 2.3 Modelling of Impinging Gas Jets on Liquid Surfaces

When a gas jet impinges on a liquid surface, it creates a depression on the surface. The depression is formed in order to balance the momentum of the gas jet with the buoyancy and surface tension force of the liquid [35]. Molloy [36] classified the depression pattern, resulting from top jetting, into three different modes: Dimpling, Splashing and Penetrating. The schematics of the cavity shapes at these three different modes are illustrated in Figure 2-11(a):

![Cavity shapes](image)

Figure 2-11 Cavity shape for (a) Top and (b) inclined jetting[37].
**Dimpling:** In this mode the jet impact velocity of the gas jet on the liquid surface is low which forms a slight surface depression. For top air jet/water system, the impact velocity is less than 15 m/s [36].

**Splashing:** With increased jet velocity and/or reduced nozzle height, a shallow depression forms on the liquid surface. Liquid droplets generate from the edge of the depression and direct outwards. For top air jet/water system, the impact velocities are in the range of 15 m/s-75 m/s [36].

**Penetration:** With further increased velocity or reduced nozzle height, much deeper penetration of the surface takes place accompanied by reduction in the amount of outwardly directed splash.

In case of inclined jetting, these three cavity modes were also observed in some later studies [37, 38]. Figure 2-11(b) shows the schematics of the cavity shapes, at three different modes, in case of inclined jetting. But the critical impact velocity on the liquid surface in case of inclined jetting should be lower than that of top jetting. This will be investigated in the present thesis.

After the introduction of supersonic oxygen lancing onto the molten iron as a method of steel refining process, a number of experimental investigation and numerical modelling were carried out by the researchers to improve the understanding of the gas jet impinging phenomena as well as to optimize both the BOF and EAF steelmaking process in terms of gas-liquid mixing, reaction kinetics and droplet generation rates. Optimization of the process is very important to improve the quality of steel as well as to reduce the operational cost. In the following section, the previous studies on gas jet impinging phenomenon are reviewed.

### 2.3.1 Experimental Modelling

#### 2.3.1.1 Depth of Penetration Studies

A number of experimental studies have been performed in the past on gas jet impinging phenomenon. Almost all of them used water to simulate the liquid steel because of its
availability and easy to handle. A schematic diagram of gas jet impinging on a liquid surface is shown in Figure 2-12:

![Schematic diagram of gas jet impinging on a liquid surface](image)

Figure 2-12 Schematic diagram of top jet impingement on a liquid surface.

Banks and Chandrasekhar [35] presented a detailed theoretical and experimental study of axisymmetric air jets impinging on water. Based on force balance between dynamic pressure and buoyancy at the impact point, they proposed a dimensionless relationship between the depth of penetration and momentum of the gas jet:

\[
\frac{M}{\rho_{lg}gh^3} = \frac{\pi}{2k_2^2} \left( \frac{p}{\rho} \right) \left( 1 + \frac{p}{\rho} \right)^2
\]

2-20

Where \( M = \frac{\pi}{4} \rho_g U_0^2 d_e^2 \) is the jet momentum, \( U_0 \) is the gas velocity \((m/s)\) at lance exit, \( d_e \) is the lance exit diameter \((m)\), \( p \) is the depth of penetration \((m)\), \( h \) is the distance \((m)\) between lance exit and liquid surface and \( k_2 = 7.9 \) is a jet constant. This equation
predicted their experimental data fairly well. Several experimental studies [39-42] also supported this relationship.

Turkdogan [43] also obtained an equation for depth of penetration from another independent theoretical analysis which was found to agree well with his experimental data. He observed in his study that although the depth of penetration decreases with increasing density of liquid, the diameter of depression remains unaffected.

Cheslak et al. [40] used fast-setting cement to determine the shape of the cavity and according to them paraboloid is a good approximation to the cavity shape. An equation for calculating the width of the cavity was proposed from the experiment. For a constant lance height, diameter of the cavity was found to be insensitive to jet momentum which was in consent with the previous investigation [35]. They also found that for a constant momentum flow rate of the jet, diameter of the cavity increases and depth of the cavity decreases if the lance height is increased. Experimental studies, carried out by Koria and Lange [44] and Hwang et al. [41] also showed similar trend. Eletribi et al. [45] reported that the equation proposed by Banks and Chandrasekhar [35], and Cheslak et al. [40], overpredicts the cavity width and depth, especially at high Reynolds number, because the theory does not properly account for the effects of the liquid viscosity and surface tension.

Koria and Lange [44] constructed a nomograph (a special graph) to determine the depth and diameter of depression from available top blowing parameter (lance height, oxygen supply pressure, diameter of nozzle and inclination angle of nozzle). They used molten iron as liquid with different carbon composition and showed that the carbon content does not affect the depth and diameter of depression. This was in agreement with the previous studies [46, 47] where it was reported that the penetration depth is not influenced by the chemical reactions at the jet impingement region.

Flinn et al. [48] used four different methods to determine the depth of penetration of high speed jet into the molten bath at non-isothermal condition. They proposed an empirical relation for predicting the penetration depth:

\[ p = 1.5 \frac{P \sqrt{d_t}}{\sqrt{\eta}} + 1.5 \]
Where $P_d$ is the dynamic pressure (psia), $h$ is the lance height (in) and $d_e$ is the nozzle throat diameter (in). When $P_d = 0$, equation (2.21) fails to predict $p = 0$. Chatterjee [49] developed an equation to predict the average depth of depression produced by supersonic jet impinging on a liquid steel:

$$\frac{p}{h} \left(1 + \frac{p-X}{h}\right)^2 = \frac{100}{\pi} \frac{M \rho_g}{\rho_gh^3 \rho_a}$$ \hspace{1cm} \text{(2.22)}$$

Where $\rho_g$ and $\rho_a$ are the density ($kg/m^3$) of gas jet and ambient atmosphere, $M$ is the momentum of (N) the gas jet and $X$ is a flow parameter which depends on the Mach number and the diameter of nozzle. The depth of penetration values calculated by equation (2.22) was found to be consistently lower than the values obtained by equation (2.21). Chatterjee [49] attributed this discrepancy to the appreciable vertical oscillations of the cavity. Also, because of the experimental setup used by Finn et al. [48], equation (2.21) calculates the maximum depth of penetration while equation (2.22) takes account the average penetration depth. From equation (2.22), Chatterjee [49] also reported that the depth of depression increases with the increase of ambient temperature at constant blowing conditions. This occurs due to the increase of potential core length of the supersonic jet at higher ambient temperatures which are described in section 2.2.

Qian et al. [42] measured the depth of depression in case of both single and two-layered liquids for different lance height and flow rate. An equation for predicting depth of depression for two stratified liquid was proposed using energy balance:

$$\frac{M}{\gamma_{mix}h^3} = \frac{\pi}{2k_2} \left(\frac{p}{h}\right) \left(1 + \frac{p}{h}\right)^2$$ \hspace{1cm} \text{(2.23)}$$

Where
Where \( L_1 \) is the height (m) of the bottom liquid layer, \( \gamma_{L_1} \) and \( \gamma_{L_2} \) are the specific weight (kg/m\(^2\)s\(^2\)) of the top and bottom liquid respectively. But this equation was found to over predict the experimental data particularly at low lance heights. In case of two liquids study, they used water/corn oil and water/kerosene to simulate metal/slag. The primary reason for using two different liquids to simulate slag was to investigate the effect of slag viscosity on the shape of the cavity. It was found that at higher slag viscosity, the oil/water interface is pulled up due to the shear driven flow. Also the entrainment of water droplets or air bubbles in the oil is lower at high oil viscosity. Turkdogan [43] observed that if the density difference between the two liquid is very high the emulsification of the two liquids does not take place.

Li and Harris [50] used water/cyclohexane to simulate metal/slag in their study and proposed an equation for predicting depth of penetration of a gas jet on two stratified liquids by doing the pressure balance:

\[
\frac{M}{\rho_{L_1}gh^3} = \frac{\pi}{2k_2} \left( \frac{L_1}{h} + \frac{\rho_{L_2} (p-L_1)}{\rho_{L_1} h} \right) \left( 1 + \frac{p}{h} \right)^2
\]

This equation was found to represent closely the experimental data when density difference between two liquids is small. When \( L_1 \) equals zero, equation (2.24) becomes the well-known equation (2.20) which was proposed for predicting depth of penetration in case of single fluid.

Meidani et al.[29] used a shrouded supersonic jet (also known as coherent jet) in their experimental study to investigate the effect of shrouding gas flow rate on the depth of penetration. Only compressed air was used for shrouding the central supersonic jet instead of combustion flame. It was concluded that depth of penetration increases with increasing shrouding gas flow rates.
Nordquist et al.[39] summarized some of the previous studies [35, 42, 44, 46, 51] on penetration depth calculation and found that the previous correlations overpredicts the depth of penetration for smaller nozzle diameters (less than 2.0mm). A new correlation was proposed based on macroscopic energy balance which was found to agree well with the experimental data for smaller nozzle diameters (0.8mm) as well as large nozzle diameters (more than 2.0mm).

None of the above mentioned study investigated the effect of lance angle on depth of depression of liquid surface. Although Koria and Lange [44] included the effect of inclined jetting, the angle of inclination was limited to 0-10° from the vertical in their experiment. This range of angle of inclination is used in the multi-nozzle lance of BOF steelmaking in order to avoid coalescences between the jets. In Electric Arc Furnace (EAF) steelmaking, supersonic oxygen jet is normally injected through the side wall at a certain angle from the vertical. Figure 2-13 shows a schematic of inclined jetting on the surface of a liquid bath.
Collins and Lubanska [52] conducted a number of experiments, long before the introduction of inclined oxygen jet in EAF steelmaking, in order to investigate the effect of jet momentum, lance angle and distance between lance exit and liquid bath on the depth of penetration. In their study, water and air were used as a liquid and gas respectively. Using dimensional analysis and experimental results, an empirical equation was proposed:

\[
\frac{M \sin \theta}{\rho_{l}gh^{1/3}} = \frac{1}{53} \left( \frac{p}{h'} \right) \left[ 1 + 19 \left( \frac{M}{\rho_{l}gh^{1/3}} \right)^{2/3} \right]
\]

Where \( \theta \) is the lance inclination angle with the horizontal surface. Other parameters of equation (2.25) have been shown in Figure 2-13. This equation fits experimental data well for lance angles over 25°. From the experimental results it was found that for a constant momentum of the jet, depth of depression decreases if the angle of inclination from the vertical is increased. Some later investigations [38, 47, 53, 54] also reported this phenomenon. This happens due to the decrease in vertical momentum of the jet with increasing lance angle. Holden and Hogg [54] showed that as the inclination angle of the jet from the vertical (liquid surface) becomes steeper, cavity depth \( (p) \) is increased and cavity length \( (b) \) is decreased, with a relatively little change in the width (cavity width perpendicular to the jet). Lingen [47] found that the volume of the cavity with inclined jetting near the bath surface is proportional to \( a^2 d e l^2 D_n \) as shown in Figure 2-13 in contrast to Collins and Lubanska [52] who reported that the volume of depression is proportional to \( p^3 \).

McGee and Irons [53] studied inclined oxygen lancing in an electric arc furnace by using a one-third scale, thin slice model of Dofasco’s Electric Arc Furnace. Distilled water and silicon oil were used to simulate steel and slag. Depth of penetration was found to decrease with increasing lance height from the bath surface and with the increase of lance angle from the vertical. The foamy slag condition was created by injecting gas through very small holes (1 mm) from the side wall. It was found that depth of penetration is higher for foaming bath condition than that in the still bath condition. This is due to the lower density of the foamed water. In order to maintain the
Dynamic similarity between the full scale furnace and thin slice model, modified Froude number similarity was used:

\[ Fr_m = \frac{\rho_g U_0^2}{\rho_l g d} \]  

Where \( \rho_g \) and \( \rho_l \) are the density \((kg/m^3)\) of gas and liquid respectively, \( U_0 \) is the velocity \((m)\) of jet at nozzle exit, \( d \) is the characteristic dimension \((m)\). If \( d \) is smaller in the one third scale thin slice model, the supersonic flow becomes subsonic in order to maintain the similar modified Froude number.

In a recent study, Solorzano-Lopez et al.[55] measured the liquid velocity using PIV (particle image velocitimity) at different lance angles. They found that the horizontal component of the liquid velocity increases with increasing lance angle from the vertical which means that the recirculation of liquid is stronger in case of highly inclined jetting.

### 2.3.1.2 Droplets Generation

When the high speed gas jet impinges on the liquid bath surface, liquid droplets are generated and ejected out of the cavity which is known as splashing. Droplet generation has both beneficial and detrimental effect. The droplets increase the surface area which in turn increases the decarburization rate. On the other hand, it causes wearing of refractories and lance tip, skulling on the converter mouth which results in reduced diameter of the furnace and hence, makes it difficult to charge raw materials. Some model studies of splashing were done by the researchers to investigate the mechanism of droplets formation as well as criteria for the onset of splashing.

Urquhart and Davenport [56] studied droplet generation study in a 1/50 scale BOF where oil and NaHCO\(_3\) solution was used to simulated to simulate slag and steel. Air was injected from the top and the jet velocity was determined using the modified Froude number (equation 2.26). It was pointed out that droplet is generated principally due to the vertical oscillation of the cavity during gas jetting. They also observed that
the horizontal displacement of the cavity results in droplet ejection but this was less frequent and more random.

Holden and Hogg [54] reported that for a given set of physical conditions (lance height, lance flow rates, lance angles), surface waves of definite amplitude and frequency are generated within the cavity. They also found that as the flow rate is increased, the frequency of the waves is also increased until they tend to break up. From physical modelling, Peaslee and Robertson [38] showed that as the velocity of the gas jet is increased, metal fingers are formed at the edge of the cavity. These metal fingers have no structural stability and are eventually torn from the cavity to produce metal droplets [38]. Three sources of metal drop formation were identified in their study [38] which are 1) tearing of the drops from the edge of the cavity or from the metal fingers, 2) tearing of small drops from in-flight drops and 3) recirculation of broken fingers or drops back into the jet mainstream. Schematic of splashing due to top and inclined jetting is shown in Figure 2-14 below:

![Schematic of splashing](image)

**Figure 2-14 Splashing of liquid due to (a) inclined jetting and (b) top jetting [37].**

In their other study, Peaslee and Robertson [5] reported that when a gas jet impinges on a liquid surface, the liquid displaced by the jet and the shearing force of the leaving gas jet generates waves inside the furnace. Figure 2-15(a) shows the formation and movement of waves during inclined jetting:
Figure 2-15 Mechanism of harmonic wave propagation (a) shows formation of the wave and (b) shows the movement of the wave [5].

During blowing, the primary wave increases in height, and both the cavity and primary wave move horizontally away from the jet. The liquid displaced by the growth of the cavity and the shear stress from high velocity jet generates the force $F_{jr}$ which increases the height of the wave. After 0.1-0.2 seconds, the original cavity moves horizontally away from the jet and a new cavity and primary wave starts forming as shown in Figure
When the gas jet no longer blows in the original cavity, the forces forming the original cavity and wave are removed resulting in an upward movement of original cavity and downward motion of primary wave. The major force trying to restore the wave and cavity to a smooth surface is the gravity \( F_g \). This completes the one cycle of oscillation of the cavity. The cavity oscillation frequency ranged from 7-8Hz in their study [5]. Koria and Lange [44] reported that the cavity oscillates at a frequency of 10-12 Hz. The magnitude of cavity depth as well as cavity diameter was found to vary by 20-25% and 10-15% of mean values respectively. McGee and Irons [53] observed cavity oscillation frequencies in the range of 6-8Hz in their water modelling study on inclined jetting. Peaslee and Robertson [38] showed that oscillation frequency does not vary greatly with the lance angle, but only with the lance heights. The frequency of oscillation was found to decrease with an increase in lance heights. Lee et al. [37] reported from their experimental study that the frequency of oscillation is independent of lance angle as long as there is no mode change of the cavity and the lance height and flow rates are constant. It was also reported that frequency of oscillation decreases with increasing lance height which is in agreement with the previous experimental study [38] on inclined jetting.

Banks and Chandrasekhara [35] observed that for top jetting splashing commences when depth of the cavity in water surface reaches a critical value of 0.014m. This is known as critical depth of depression. Some later investigations also [8, 40, 43, 51] reported the similar criterion for the onset of splashing.

Chatterjee and Bradshaw [51] carried out numerous room-temperature experiments to investigate the effect of liquid properties, gas properties and lance height on the critical depth of depression. From their studies it was found that critical depth of depression is independent of gas properties, slightly influenced by the lance height but solely dependent on liquid properties. With the aid of dimensional analysis, an equation was proposed to calculate the critical depth of depression of liquid with different properties:

\[
\left( \frac{\rho_1}{\sigma} \right)^{1/2} = 0.531 \log \left( \frac{\mu_1}{\rho_1 \sigma^2} \right) + 11.33
\]
They also measured the volume of liquid splashed during top blowing by collecting the liquid droplets in a perspex tray. The tray was divided into five compartments in order to investigate the radial distribution of splashing as shown in Figure 2-16.

Figure 2-16 Perspex tray for measuring radial distribution of liquid splashed just above the bath surface [51].

Figure 2-17 shows that splashing intensity is highest in the immediate region surrounding the cavity and decreases with increasing radial distance from the impingement point. It was also reported from their cold modelling study that splashing rate increases with decreasing lance height. Experiments performed by Turner and Jahanshahi [57] also showed this phenomenon.

Standish and He [58] measured the volume of splashing by measuring the total amount of splash ejected out of the vessel during jetting. Their study showed that splashing rate is not a monotonous function of lance height. The splashing rate increases with lowering the lance height up to certain point and after that splashing rate decreases with decreasing lance height as shown in Figure 2-18. This occurs due to the transition from splashing mode to penetrating mode as described by Molloy [36]. They [58] also
observed that the proportion of large droplets increases with increasing top gas flow rate. At a constant top gas flow rate, splashing rate was found to increase with bottom gas flow rate [57, 58].

Figure 2-17 Influence of jet momentum on radial distribution of droplets [51].

Figure 2-18 Variation of droplet generation rate with lance heights. [58].
Paul and ghosh [59] measured splashing by connecting one terminal of a DC supply with a steel wire net on the wall end and Cu rings around the vessel. The other terminal was connected to a copper rod dipped into the liquid. Upon agitation water made contact with the steel net/copper rod and hence closed the circuit to record splashing data on an X-Y recorder. To obtain a qualitative data of spitting, a sponge was used above the liquid surface. The difference in weights of the sponge before and after the experiment was taken as the amount of spitting.

Splashing studies [8, 51, 58, 59] that have been described so far were only focused on the total volume of liquid splashed during the blow. But knowledge of splashing pattern on the wall of the furnace is very important to locate the area vulnerable to refractory wear inside the furnace. In an attempt to obtain the splashing distribution on the wall, Garg and Peaslee [60] collected the splashed liquid through 67 small holes in their cold modelling experiment. The holes covered one-sixth of the model wall circumference and were connected to splash collecting containers through rubber pipes. The splashed liquid was collected and measured at different operating conditions (lance height, flow rate, viscosity, number of nozzle, lance angle). It was concluded that the gas flow rate has the major effect on splashing rate. Luomala et al.[61] collected splashed droplets from the whole model wall circumference using 60 holes in their cold modelling experiment. Their results showed that splashing is highest on the lower part of the vessel and introduction of bottom blowing increases the intensity of splashing which is in consent with the previous studies [6].

In the experimental study, performed by He and Standish [6], water and mercury were used to simulate liquid steel and droplet generation rates were measured for each case. In order to compare the results obtained from the two different experiments, a dimensionless Nominal Weber number was proposed which expresses a ratio of the momentum intensity to the main liquid properties:

\[
N_{we} = \frac{\rho \sqrt{\frac{\Delta z}{g \delta}}}{\sqrt{\frac{\rho_l g \sigma}}}
\]
Figure 2-19 shows that the droplet generation starts after a critical Nominal Weber number is reached which is independent of the liquid properties.

![Figure 2-19 Comparison of the results from two modelling experiments [6].](image)

Li and Harris [50] pointed out that the onset of splashing is related to the instability at the gas-liquid interface. When two stratified phases are in relative motion, the interface between the two phases may not be stable and small perturbation on the interface can
result in large waves on the interface under certain circumstances. This is known as Kelvin–Helmholtz instability. Figure 2-20 shows two different fluid layers having different densities, \( \rho_1 \) and \( \rho_2 \), flowing horizontally with velocities \( U_1 \) and \( U_2 \). From their calculation it is shown that the interface is stable if the relative velocity is sufficiently small to satisfy the following equation:

\[
(U_1 - U_2)^2 < \frac{2(\rho_1 + \rho_2)}{\rho_1 \rho_2} \sqrt{\sigma g (\rho_1 - \rho_2)}
\]  

2-29

Where \( \sigma \) is the surface tension (N/m). If the onset of splashing is controlled by interface instability, the critical gas jet velocity for the onset of splashing can be obtained from equation (2.29) by assuming \( \rho_1 = \rho_t \), \( \rho_2 = \rho_g < \rho_t \), \( U_2 = U_g \), and \( U_1 = U_l < U_G \):

\[
\frac{1}{2} \rho_g U_g^2 = \sqrt{\sigma g \rho_t}
\]  

2-30

Where \( U_g \) is the critical tangential gas velocity for the onset of splashing. The critical tangential velocity \( (U_g) \) is related with free turbulent jet axial velocity \( (U_j) \) by the following equation:

\[
U_g = \eta U_j
\]  

2-31

Where \( \eta = 0.44 \) is an experimentally determined constant. Subagyo et al.[7] defined a dimensionless Blowing number \( (N_g) \) in their droplet generation study by rearranging equation (2.30):
The droplet generation starts when Blowing number exceeds unity because in that case the left hand side of equation (2.29) becomes higher than the right hand side and hence, the interface becomes unstable. An empirical equation for calculating the droplet generation rate was proposed [7]:

$$N_B = \frac{\rho_g U_0^2}{2\sqrt{\sigma g \rho_l}}$$  \hspace{1cm} 2-32

2-33

$$\frac{R_B}{F_g} = \frac{(N_B)^{3.2}}{[2.6 \times 10^6 + 2.0 \times 10^{-4}(N_B)^2]^{0.2}}$$

Where $R_B$ is droplets generation rate (kg/s) and $F_g$ is volumetric gas flow rate (Nm$^3$/s). Figure 2-21 shows the variation of droplet generation rates with Blowing number:

Figure 2-21 Effect of Blowing number on the generation rate of metal droplets [7].
The graph shows that droplet starts to generate when the Blowing number equals to unity. Subagyo et al. [7] showed that equation (2.33) is valid for both cold and hot systems. From Figure 2-19 and 2-21, it is evident that both the Nominal Weber number and Blowing number indicates the onset of splashing at some critical value. But the Blowing number provides some physical sense because it indicates how many times the Kelvin-Helmholtz instability criterion is exceeded.

All of the above mentioned studies on droplet generation rate have been done for the top jetting only. In EAF steelmaking, the oxygen jet is injected at a certain angle from the vertical. Patjoshi and co-workers [8] studied the amount of splashing by measuring the difference in volume of the liquid bath before and after the blow for inclined jetting. Three different lance angles $5^0$, $8^0$ and $12.5^0$ from the vertical were used in their experiment. It was found that the splashing rate increases with the increase in lance inclination angle. This small range of inclination angles are used to prevent the coalescence of multiple jets in multi-nozzle top jetting. In EAF steelmaking, the angle of inclination is higher than $12.5^0$. Peaslee and Robertson [38] reported that angle greater than $45^0$ can cause excessive forward splashing on the furnace wall and angles less than 20 degree can result in back splashing to the lance tip and also bottom wear due to high penetration depth. McGee and Irons [53] injected air on water at four different lance angles: $20^0$, $35^0$, $40^0$ and $45^0$ from the vertical. From visual observations, it was reported that a lance angle of $40^0$ from the vertical produce least amount of splashing. Also foamy bath condition was found to reduce the splashing rate. But no qualitative measurements were made on the amount of splash generated.

2.3.2 Numerical Approach

As discussed earlier, the experimental studies are generally carried out using cold models or small scale furnaces which are designed using dimensional analysis. But it is not always possible to maintain complete similarity in experimental model with the real system. In some cases, experimental studies involve high costs and also require long time to build the experimental set up. Therefore, researchers are trying relentlessly to develop reliable and efficient modelling techniques to model the jet impingement.
phenomenon. In this section, a literature review on the numerical modelling of gas jet impinging phenomenon has been presented.

Early on some analytical modelling studies [62, 63] of the jet impingement phenomenon were carried out by the researchers to predict the cavity shape. Olmstead and Raynor [62] formulated a non-linear singular integral equation, using conformal mapping and finite Hilbert transforms, to predict the depression on liquid surface created by a gas jet. The integral equation was approximated by a set of non-linear algebraic equations which were solved numerically by a method of repeated linear corrections. But there approach was limited to small angle depression of liquid surface.

Rosler and Stewart [63] proposed a second order differential equation using force balance at the liquid surface which, upon integration, describes a steady state indentation profile. They assumed that the pressure distribution on the liquid surface is similar to that on a flat plate. Figure 2-22 shows the schematics of the indentation profile and the force balance.

Figure 2-22 Schematics of indentation profile and force balance [63].
From this figure, the second order differential equation, describing the shape of the cavity can be written as follows:

\[
\frac{d^2x}{dh^2} + \frac{1}{x} \frac{dx}{dr} \left[ 1 + \left( \frac{dx}{dr} \right)^2 \right] = \frac{1}{\sigma} \left[ (\rho_1 - \rho_g) g x - \Delta P - \tau \frac{dx}{dr} \right] \left[ 1 + \left( \frac{dx}{dr} \right)^2 \right]^{\frac{3}{2}}
\]

Where \( x \) and \( r \) are the coordinates shown in Figure 2-22, \( \Delta P \) is the pressure difference across the interface and \( \tau \) is the interfacial shear stress \( (N/m^2) \) caused by the impinging jet flow. But it was found that the application of this differential equation is limited to stable non-oscillating cavity.

Evestedt and Medvedev [64] used image processing tools and edge detection technique together with the force balance model of Rosler and Stewart [63] in order to describe the cavity profile. They concluded that cavity oscillates vertically and the assumption of steady non-oscillating cavity leads to large variance of the depth and diameter estimates.

With the advent of modern computational techniques and resources, the numerical modelling approach is becoming more reliable and efficient in terms of time and costs. Szekeley and Asai [65] carried out the first computational fluid dynamics model on jet impingement. They assumed a constant shape of the cavity based on the available experimental observations and calculated the flow field in liquid caused by an impinging air jet. They computed the velocity, vorticity and turbulent kinetic energy distributions inside the liquid bath. The computed results showed that the turbulent kinetic energy is relatively low in the bulk of the liquid phase as shown in Figure 2-23. A reasonable good agreement was reported between the calculated and measured liquid velocity.

Qian et al [66] developed a numerical model to predict the interface deformation, created by the impinging gas jet on a liquid bath, along with the liquid flow field and temperature distribution. A heated liquid bath was considered in their study. The gas phase flow field was calculated first assuming a flat interface and then the interface shape was obtained by considering the pressure at the gas side on the interface. The liquid flow field was then calculated with the new interface shape and shear stress at the
interface. The final solution was obtained via an iterative process. The predicted results of the interface shape compared well with experimental data for low flow rate. However, at higher flow rates there was some discrepancy between calculated and experimental data.

Olivares et al.[67] carried out both physical modelling and mathematical simulation of gas-liquid interactions. In the numerical modelling, VOF [68] method was used to calculate the sharp gas-liquid interface. The crater and surface waves formed by the gas jet were well predicted by their numerical model. However, the cavity dimensions were not reported from the numerical simulation.

Nakazono et al.[69] developed a CFD model of supersonic jet impinging on high temperature liquid metal. The simulation was done for single phase (gas phase) only. Liquid phase was avoided by assuming a fixed cavity geometry which was determined from the balance of impact pressure, shear stress, surface tension and buoyancy of the liquid. The reactions in the gas phase and on liquid surface were included in their study. Surface reactions were found to have very negligible effect on the cavity geometry which is in consent with the previous experimental studies [44, 46].

Figure 2-23 The computed turbulent kinetic energy profile($cm^2/s^2$) [65].
Odenthal et al. [70] simulated supersonic oxygen jet impingement onto a slag and molten iron bath in a 335 ton furnace at non-isothermal condition. The supersonic jet was assumed compressible at Ma=2.21 because the simultaneous solution of compressible gas jet in conjunction with incompressible liquid flow is still a big challenge in the field of CFD. The CFD simulation was found to under predict the cavity depth and diameter when compared with calculated cavity dimensions from the equation proposed by Koria and Lange [44]. Splashing of liquid was observed but not quantified.

Nguyen and Evans [71] included the interfacial surface tension effect in their jet-liquid interaction model via the Laplace equation which are discussed in Chapter 4. The VOF method was used in their simulation to calculate the gas liquid interface. During calculation, gas-liquid interface was updated in every interface cell by using the piecewise linear interface construction (PLIC) method [72]. The influence of furnace diameter on the deformation of the gas-liquid interface was investigated. Cavity dimensions obtained from their CFD simulations were found to agree well with the theoretical predictions for the unconfined system. In case of confined system, CFD model under predicted the cavity dimensions. They attributed splashing of liquid out of the vessel for the under prediction.

Liovic et al. [73] used the Continuum Surface Force (CSF) model [74] to incorporate the surface tension effect at the gas-liquid interface for different metallurgical flows. Their model simulated the formation of droplets in case of top submerged gas injection but no qualitative measurements were done.

The CSF model and the laplace equation used by Nguyen and Evan [71] was based on simplified surface boundary conditions where shear stress terms were completely neglected. Recently, Hwang and Iron [75] applied full stress boundary condition at the gas liquid interface to transfer the gas momentum properly to the liquid surface. Figure 2-24 shows the calculated surface profile variations with time at 0.12m lance height and 0.05Nm³/min flow rates. Their figure shows that the full stress boundary conditions at the gas liquid interface can predict the waves generated in the surface more realistically. The calculated depth of penetration was found to increase with increasing jet flow rates but decrease with increasing lance height as shown in Figure 2-25 which is in agreement with previous experimental studies [35, 40, 42].
Figure 2-24 Surface profile variations with time [75].

Figure 2-25 Variation in mean depth of penetration with changing lance heights and flow rates [75].
Ersson et al. [76] reported that turbulence modelling has a significant impact on the prediction of penetration depth. The standard \( k - \varepsilon \) [14] turbulence model was found to underpredict the penetration depth when compared with the analytical equation proposed by Banks and Chandrasekhara [35]. They proposed a modification of the constant \( C_{\varepsilon_2} \) of standard \( k - \varepsilon \) model by reducing the constant value from 1.92 to 1.78 in order to get a better prediction of the penetration depth. However, it was also reported from their study that the change in \( C_{\varepsilon_2} \) value changes the flow pattern of the liquid phase. In their another study, Ersson et al. [77] coupled CFD software with the thermodynamic database to simulate the reactions between gas-steel, steel-slag and gas-slag along with the simulation of jet impingement phenomenon.

Besides top jetting, some mathematical and numerical simulations of inclined gas injection were also available in the literature. Gu and Irons [78] numerically simulated the inclined jet-water interaction of a 1/3 scale “thin-slice” EAF model. In their simulation, the free surface modelling of the liquid were done using the Volume of Fluid (VOF) technique [68]. The computed depth of cavity, liquid phase velocity and gas phase velocity were comparable with the experimental data for low flow rate of gas jet (0.57Nm\(^3\)/min). At higher gas flow rates, the simulation experienced rapid divergence. They attributed splashing of liquid droplets for the divergence.

McGee and Iron [53] developed a mathematical model to calculate the penetration depth at different lance heights, lance angles and flow rates. A number assumptions were made to make the model simpler which are listed in their paper [53]. The penetration depth was found to have parabolic relationship with flow rates and a linear relationship with lance angle. They also developed a simple model for predicting the depth of penetration by balancing the vertical jet momentum against the buoyancy forces of the displaced liquid:

\[
L = \frac{3 \rho g}{\rho g} \left( \frac{U_0^2}{A_0} \right) \cos \alpha
\]

Where \( L \) is the summation of axial lance height and penetration depth, \( A_0 \) is the lance cross-sectional area (\( m^2 \)) and \( U_0 \) is the exit velocity of the gas. The equation shows that...
penetration depth depends on the square of the lance flow rate and varies linearly with lance angle.

Figure 2-26 Variation of depth of penetration with flow rates[53].

Figure 2-27 Variation of depth of penetration with lance angles[53].
Recently, Solorzano-Lopez et al.[79] developed a numerical model of inclined jetting on water surface. The computed cavity dimensions were compared with their experimental data and were found to be in good agreement. The liquid free surface velocity was found to increase with increasing lance flow rates, decreasing lance heights and increasing lance angles from the vertical. The higher liquid surface velocity indicates stronger mixing inside the bath. The computed liquid flow patterns near the cavity agreed reasonably well in magnitude but did not agree well in trend with experimentally determined flow patterns. Splashing of the liquid phase was not modelled although they reported some perturbations and instabilities of the cavity. He et al.[80] carried out numerical simulation of a 150 ton EAF converter at steelmaking temperature using three phases: gas, slag and steel. The simulated depth of penetration was validated against an empirical equation which was developed from the water modelling study. The water model was designed using the Froude number similarity with the 150 ton converter. The calculated penetration depth was found to underpredict the depth of penetration magnitude calculated by empirical equation. It was argued that the underprediction in numerical model resulted from the slag layer on the molten iron whereas the empirical equation was developed from the water modelling study where no slag layer was considered.

Guo et al.[81] performed mathematical modelling of bottom gas injection into a one third scale thin slice model of a steelmaking furnace. Navier- Stokes equation had been solved using both Lagrangian framework and Eulerian approach. Eulerian methods consider that the two phases consist of interpenetrating flow field, so that a set of equations is solved for each phase. In contrast, Lagrangian methods trace the motion of the discrete phase (bubbles in this case) through the continuous liquid phase. The plume position of the gas bubbles from the bottom of the model furnace was calculated. A reasonable agreement between the calculated data and the experimental results was observed. In some other studies, Gu and Irons [82, 83] reported that Lagrangian model provides better representation of the fluid flow for the bottom injection and foaming gas evaluation compared with Eulerian framework.
2.4 Research Objectives

Based on the literature review, the following goals of the current research have been set to contribute to the understanding of the supersonic jet behaviour, jet liquid interactions and droplet generation rate inside the steelmaking furnace. The computational fluid dynamics (CFD) modelling approach are used in the present study for investigating these phenomena. Models provide powerful tools for making useful predictions, developing a theoretical understanding of the system, and provide a framework to advance the current understanding of the process. Also, the complexity of steelmaking and problems associated with measuring and visualizing the phenomenon being studied at steelmaking temperatures, necessitates the use of different computational models. Along with the CFD modelling, an experimental study will also be carried out to validate the developed CFD model.

In the present study, first of all, a numerical simulation of the supersonic jet, at different ambient temperatures, have been carried out and validated against the experimental data. It is shown that the \( k-\varepsilon \) turbulence model with compressibility correction underpredicts the potential core length of the jet at high ambient temperature when compared with the experimental data of Sumi et al.[22]. The reasons for this underprediction have been investigated. A modification of the \( k - \varepsilon \) model has been proposed for predicting the potential core length, axial velocity and temperature distributions of the supersonic jets more accurately at steelmaking temperatures.

The numerical model is then extended for modelling the coherent supersonic jets. From the literature review on coherent supersonic jets, it is evident that the existing literature still lacks a comprehensive description on how the shrouding combustion flame affects the supersonic jet characteristics. In the present study, computational fluid dynamics model of supersonic jet with and without combustion flame has been developed and validated against the available experimental data. Then the effect of combustion flame on turbulent shear stress region, vorticity region of the supersonic jet has been discussed for the purpose of a clear understanding of how the technology works. This knowledge is important for further improvement of the technology.

After that, the modelling of the supersonic gas jet impingement on a liquid surface have been performed. It was found from the literature review that all the jet-liquid interaction
modelling was done either at low velocity of the jet (less than 100 m/s) or assuming the supersonic jet incompressible. The modelling of compressible jet impingement on the incompressible liquid surface is still a significant challenge in CFD. In the present study, a new approach for modelling the shrouded supersonic jet impingement on a liquid free surface has been presented using two different computational domains. The existing literature also showed that the droplet generation rate decreases when cavity operates in the penetrating mode. The contributing factors for the reduction in droplet generation rate during penetrating mode are investigated from the CFD model. The limitations of dimensionless Blowing number theory in calculating the droplet generation rate during the penetrating cavity mode have been discussed.

The experimental studies on droplet generation rate, available in the literature, were carried out for top jetting only. In the present study, an experimental study has been carried out to investigate the effect of lance inclination angle, lance heights and flow rates on the cavity dimensions and droplet generation rates. The Blowing number theory of Subagyo et al.[7] was originally proposed for top jetting only. In order to calculate the Blowing number in case of inclined jetting, a modification of the Blowing number theory is proposed, by including the lance inclination angle, from the current experimental study.

Finally, CFD modelling of inclined jetting on the liquid surface has been carried out and validated against the present experimental study. Effort has been made to numerically simulate the droplet generation rate, and the difficulties involved in the simulation are discussed.

2.5 Brief Overview of Computational Fluid Dynamics Modelling

The development of high speed digital computer during the twentieth century has had a great impact on the way how the principles of fluid mechanics and heat transfer are applied for solving the engineering problems. Over the past half century, we have witnessed the rise of a new methodology for solving the complex problems in fluid mechanics and heat transfer. This new methodology has become known as Computational Fluid Dynamics (CFD). Increasingly CFD is becoming a vital
component in the design of industrial products and processes. However, CFD is not a substitution of experimental work but a very powerful additional problem solving tool. Validation of CFD results must be done by comparing it with some form of experimental work. It is particularly useful when focus is on improving the current technology by optimizing present operating conditions. It can provide detailed information on various important parameters under different operating conditions [84].

In Computational Fluid Dynamics Modelling, the flow related phenomenon such as temperature, pressure, velocity and density are described by a set of partial differential equations (PDE). These partial differential equations can be written in a general form as follow [84]:

\[
\frac{\partial (\rho \Phi)}{\partial t} + \text{div}(\rho \Phi U) = \text{div}(\Gamma \text{grad}\Phi) + S_{\Phi}
\]  

2-36

Where \( \rho \) is the density \((kg/m^3)\) and \( U (m/s) \) is the velocity of the flow in vector form. This equation is called the transport equation for property \( \Phi \) showing the different transport processes: unsteady term and convection term on the left hand side and diffusive term and source term on the right hand side respectively. By setting \( \Phi \) equal to 1, \( U, V, W \) and \( T \), and selecting appropriate values for the diffusion coefficient and source terms, the equation (2.36) will result in mass, momentum and energy conservation equations which will be described in detail in the next chapter.

The analytical solution of these PDE’s is extremely difficult except in some simple cases. In order to obtain an approximate solution, numerical approach is used where the PDE’s are represented by a set of algebraic equations and solved by computer iteratively. The PDE’s are converted to algebraic equations by using different discretization methods which are described in next section.
2.5.1 Discretization Method

After selecting the appropriate PDE’s for a particular problem, the equations are discretised at some discrete locations in space and time and a set of algebraic equations are obtained. There are many discretization methods available in the literature but three of them are widely used: Finite difference (FD), Finite volume (FV) and Finite element (FE) methods.

2.5.1.1 Finite Difference Method

In Finite Difference (FD) method, the starting point is the conservation equation in differential form. The differential equation is approximated at each grid point by replacing the partial derivatives by approximations in terms of the nodal values of the functions. The result is one algebraic equation per grid node, in which the variable value at that node and a certain number of neighbour nodes appear as unknowns [85]. Taylor series expansion or polynomial fitting is used to obtain approximations to the first and second derivatives with respect to the coordinates. When necessary, these methods are also used to obtain variable values at locations other than grid nodes (interpolation). On structured grids, the FD method is very simple and effective. The disadvantage of FD methods is that the conservation is not enforced unless special care is taken. Also, the restriction to simple geometries is a significant disadvantage in complex flows [85].

2.5.1.2 Finite Volume Method

The Finite volume (FV) method uses the integral form of the conservation equations as its starting point. The solution domain is subdivided into a finite number of contiguous control volumes and the conservation equations are applied to each control volume (CV). At the centroid of each CV lies a computational node at which the variable values are to be calculated. Interpolation is used to express variable values at the CV surface in terms of nodal values. Surface and volume integrals are approximated using suitable quadrature formulae. As a result one obtains an algebraic equation for each CV, in which a number of neighbour nodal values appear. The FV method is suitable for
complex geometries and is conservative. This method is simplest to understand and to
program. The disadvantage of FV method compared to FD method is that methods of
order higher than second are more difficult to develop in 3D [85].

2.5.1.3 Finite Element Method

In Finite Element (FE) method, the domain is subdivided into a set of discrete volumes
or finite elements that are generally unstructured. This method is similar to FV method
in many ways. The distinguishing feature of FE method is that the equations are
multiplied by weight function before they are integrated over entire domain. In simplest
FE methods, the solution is approximated by a linear shape function within each
element in a way that guarantees continuity of the solution across element boundaries.
This approximation is then substituted into the integral of the conservation law and the
equations to be solved are derived by requiring the derivative of the integral with
respect to each nodal value to be zero; this corresponds to selecting the best solution
within the set of allowed functions (the one with minimum residual). The result is a set
of non-linear algebraic equations. The advantage of the FE method is the ability to deal
with complex geometries and easy to analyse mathematically. The principal drawback is
the use of unstructured grid. The matrices of the linearized equations are not as well
structured as those for regular grids making it more difficult to find efficient solution
methods [85].
Chapter 3

3. Numerical Modelling of Supersonic Jet


3.1 Introduction

This chapter describes the fundamental governing equations of fluid flow and the methods used to numerically solving the equations. An overview of turbulence modelling techniques currently available in the literature has been provided and a modification of the $k - \varepsilon$ turbulence model is proposed for modelling the supersonic jet behaviour at steelmaking temperature. Then the numerical modelling of supersonic free jets and coherent jet is described in detail.

3.2 Governing Equations

The governing equations of the computational fluid dynamics are the mathematical statements of three fundamental physical principles [85]:

1. Mass is conserved
2. Newton’s second law, $F = mg$.
3. Energy is conserved.

Using these three fundamental physical principles along with Reynolds averaging technique for turbulent fluid flow, one can arrive in the following Reynolds Averaged Navier-Stokes (RANS) equations [84]:

- Mass conservation equation

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_i}{\partial x_i} = 0
\]

Where $\rho$ is the density of the fluid($kg/m^3$), $U_i$ is the mean velocity($m/s$) component in the $i$th direction.

- Momentum conservation equation
\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial \left(\tau_{ij} - \rho u_i u_j\right)}{\partial x_j} + \rho g_i \tag{3-2}
\]

\[
\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij}\right) \tag{3-3}
\]

Where \( P \) is the pressure of fluid\( (Pa) \), \( \tau_{ij} \) is viscous stress\( (N/m^2) \), \( u_i, u_j \) are the fluctuating velocity \( (m/s) \) component in \( i \)th and \( j \)th direction respectively and \( \mu \) is the molecular viscosity\( (N - s/m^2) \). \(-\rho u_i u_j\) is known as “Reynolds stresses” and is used to represent the effect of turbulence. Modelling of the Reynolds stresses have been described in next section.

- Energy conservation equation

\[
\frac{\partial \rho H}{\partial t} + \frac{\partial (\rho H u_i)}{\partial x_i} = -\frac{\partial}{\partial x_i} \left( q_i + c_p \rho u_i t' \right) + \frac{\partial}{\partial x_i} \left( \tau_{ij} u_j - \rho u_i u_j u_j \right) + \frac{\partial P}{\partial t} \tag{3-4}
\]

Where \( H \) is the total enthalpy \( (J/kg) \), \( c_p \) is the specific heat at constant pressure \( (J/kg^{-1}K^{-1}) \) and \( t' \) is the fluctuating component of temperature. Heat transferred by conduction \( (W/m^2) \), \( q_i \) is:

\[
q_i = K \frac{\partial T}{\partial x_i} \tag{3-5}
\]

Where \( K \) is the thermal conductivity of fluid \( (W/m^{-1}K^{-1}) \). The term \( \rho u_i t' \) is known as turbulent heat flux and is modelled as:

\[
\rho u_i t' = \frac{\mu_t}{\partial x_i} \frac{\partial T}{\partial x_i} \tag{3-6}
\]
Where $Pr_t$ is the turbulent Prandtl number. Most common values of turbulent Prandtl number is 0.9 and it is satisfactory for shock free flows up to low supersonic speeds and low heat transfer rate [86]. When a cold supersonic jet (300K) is introduced inside a steelmaking furnace of around 1800K ambient temperature, the heat transfer rate from the hot environment to the cold jet will be very high due to large temperature gradient. Wilcox [86] recommended to use $Pr_t=0.5$ for free shear flow and high heat transfer problems. Hence, $Pr_t=0.5$ was used in this simulation. The second term and third term on the right hand side of equation (3.4) is the viscous heating and pressure work respectively.

Hence, the motion of a fluid in three dimensions can be described by a system of five partial differential equations: mass conservation (3.1), x, y and z momentum equation (3.2) and energy equation (3.4). The number of unknown variables is six: density, three velocity components, pressure and temperature. If the density is constant and there is no heat transfer in the system, then the flow field can be solved by using equation (3.1) and (3.2). But in case of compressible flow and the problems involving heat transfer, a sixth equation, known as equation of state, is used to calculate the six unknown variables. The equation of state provides a thermodynamic relation between pressure temperature and density [84]:

$$P = \rho RT$$  \hspace{1cm} \text{(3.7)}

Where $R$ is the gas constant (Jkg$^{-1}$K$^{-1}$). The equation (3.7) is also known as perfect gas laws. In a system where the density is not constant, equation (3.7) is used to calculate the density of the fluid after computing the pressure, velocity components and temperature from the mass, momentum and energy equations.
3.3 Turbulence Modelling

In steelmaking, the velocity of the oxygen jet is very high. A chaotic and random state of motion develops at such high velocity in which the velocity and other flow properties change continuously with time. This regime is known as turbulent flow. A typical point velocity measurement might exhibit the form shown in Figure 3-1 where $\varphi(t)$ is the fluctuating turbulent velocity with time and $\Phi$ is the steady mean velocity.

![Figure 3-1 Typical point velocity measurement in turbulent flow [84].](image_url)

When the high speed gas jet enters into the region of stationary fluid, a mixing layer is formed at the interface of high speed gas jet and surrounding fluids which in turn slows down the gas jet. This occurs due to the turbulent nature of the high speed gas particle which causes vigorous mixing of adjacent fluid layers and rapid widening of the region across which velocity change takes place. Turbulent fluctuations are three dimensional and have a wide range of time and length scales. A substantial amount of research effort is dedicated to the development of numerical methods to capture the important effects due to turbulence. These methods can be grouped in following three categories [86]:

- Category 1
- Category 2
- Category 3
3.3.1 Direct Numerical Simulation

In direct numerical simulation (DNS), the time dependent Navier-Stokes equations without averaging are solved on a sufficiently fine spatial mesh with sufficiently small time steps to resolve Kolmogorov length scales at which energy dissipation takes place and fastest fluctuation. Using DNS method, precise details of turbulence parameters and their transport at any point in the flow can be calculated. But this method is very costly in terms of computing resources, so the method is not used for common industrial flow computation [86].

3.3.2 Large Eddy Simulation (LES)

This is a different approach of computation of turbulent flows where the larger eddies, which interact with and extract energy from the mean flow, are computed with a time dependent simulation and the smaller eddies, on the other hand, are computed by means of a sub-grid scale (SGS) model. In LES method, a spatial filtering operation is done on the unsteady Navier-Stokes equation prior to computation to separate the larger and smaller eddies. During spatial operation, information relating to the smaller, filtered out turbulent eddies is destroyed. The effect of the smaller, unresolved eddies on the larger, resolved eddies are described by means of a SGS model. LES also requires large amounts of computer time and storage but the use of this technique, to address industrially relevant CFD problems with complex geometry, is increasing with the advent of powerful computing resources [86].

3.3.3 Reynolds-Averaged Navier-Stokes (RANS) Model

For most Engineering purposes it is unnecessary to resolve the details of the turbulent fluctuation which is also computationally expensive. To avoid the direct calculations of small-scale turbulence, in most engineering application the turbulent flow computations are done by time averaging the Navier-Stokes equations. This is called the Reynolds-Averaged Navier-Stokes equations because it is based on ideas proposed by Osborne Reynolds over a century ago. In this approach, the instantaneous flow property $\hat{\varphi}$ is
decomposed into a mean $\Phi$ and fluctuations around the mean $\varphi$ as shown in Figure 3-1 [84]:

$$\hat{\varphi} = \Phi + \varphi$$

Substitution of the dependent flow variables in Navier-Stokes equations by equation (3.8), results the RANS equations. The averaging process results in six extra stresses in time averaged momentum equations:

$$-\rho \bar{u}_i^2, \ -\rho \bar{u}_j^2, \ -\rho \bar{u}_k^2, \ -\rho \bar{u}_i \bar{u}_j, \ -\rho \bar{u}_j \bar{u}_k, \ -\rho \bar{u}_i \bar{u}_k.$$

These are known as Reynolds stress. The presence of turbulent Reynolds stresses in the $x$, $y$ and $z$ momentum equations means that the latter are not closed set. A number of turbulence models are available in the literature to predict the Reynolds stresses and close the RANS equations which are presented in Table 3-1 based on the additional transport equations that need to be solved along with RANS equations:

Table 3-1 Different types of turbulence model [84].

<table>
<thead>
<tr>
<th>No of extra transport equations</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>Zero</td>
<td>Mixing length model</td>
</tr>
<tr>
<td>One</td>
<td>Spalarat-Allmaras model</td>
</tr>
<tr>
<td>Two</td>
<td>$k - \epsilon$ model</td>
</tr>
<tr>
<td></td>
<td>$k - \omega$ model</td>
</tr>
<tr>
<td>Seven</td>
<td>Reynolds Stress model</td>
</tr>
</tbody>
</table>

Of the tabulated models, the zero equation and two equation models are based on Boussinesq approximation which states that Reynolds Stresses are proportional to mean rates of deformation:

$$-\rho \bar{u}_i \bar{u}_j = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} (\rho k + \mu_t \frac{\partial u_k}{\partial x_k}) \delta_{ij}$$

$$3-9$$

Where $\mu_t$ is the turbulent viscosity($N - s/m^2$), $k$ is the turbulent kinetic energy($m^2/s^2$) and $\delta_{ij}$ is the Kronecker delta. The underlying assumption of this approximation is that the turbulence is isotropic [84]. Turbulent viscosity is the product
of turbulent velocity scale and turbulent length scale. In “mixing length” model the turbulent velocity scale is directly related to the magnitude of mean shear and the length scale is empirically described. No extra equation is needed to compute the Reynolds stresses. That is why it is known as the zero equation model [84].

In the two equation model, two different transport equations are solved: one for the turbulent velocity scale related variable and the other for length scale related variable. All two equation models use turbulent kinetic energy as the velocity scale related variable and solve a transport equation for turbulent kinetic energy. But different length scale related variables are used in different models. In the \( k - \varepsilon \) model, rate of turbulent energy dissipation \( \varepsilon \) is used as a length scale related variable but in \( k - \omega \) model, the turbulence frequency \( \omega \) is used [84, 86, 87].

In the one equation model, a single transport equation for turbulent kinetic energy is solved to obtain the turbulence velocity scale and the length scale is defined by using empirical formula. In the Spalart-Allmaras one equation model, a transport equation for turbulent kinematic viscosity is solved and then by using an empirical formula the turbulent eddy viscosity is calculated [86].

In the Reynolds Stress Model (RSM), the Boussinesq approximation of equation (3.9) is discarded and all the individual Reynolds stresses are solved by separate transport equations. The six transport equations, one for each Reynolds stress, contain pressure strain and dissipation terms whose individual effects are unknown and cannot be measured. Assumptions are made about these unknown terms, and the resulting PDEs are solved in conjunction with the transport equation for turbulent dissipation rate \( \varepsilon \) [84].

### 3.3.4 Turbulence Model Used in The Present Study

In the present study, two equation \( k - \varepsilon \) turbulence model [14] was used to close the RANS equations. In this model the turbulent kinetic energy \( k \ (m^2/s^2) \) and the dissipation rate \( \varepsilon \ (m^2/s^3) \) are obtained from the following transport equations:

- Kinetic energy equation:
\[ \frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_j k)}{\partial x_j} = -\rho \frac{\partial U_i}{\partial x_j} \frac{\partial U_i}{\partial x_j} \frac{\partial}{\partial x_j} - \frac{\mu_t}{\rho \sigma} \frac{\partial \rho}{\partial x_j} g_i \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] - \rho \varepsilon \]  

\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_j \varepsilon)}{\partial x_j} = \left( -C_{\varepsilon 1} \rho \frac{\partial U_i}{\partial x_j} - C_{\varepsilon 3} \frac{\mu_t}{\rho \sigma} \frac{\partial \rho}{\partial x_j} g_i + C_{\varepsilon 4} \frac{\partial U_K}{\partial x_K} \right) \frac{\varepsilon}{k} \]

\[ + \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} \]

Where \( C_{\varepsilon 1}, C_{\varepsilon 2}, C_{\varepsilon 3}, C_{\varepsilon 4}, \sigma_k, \sigma_\varepsilon, \sigma_p \) are the constants for the \( k - \varepsilon \) model and their values are 1.44, 1.92, 0.8, 0.33, 1.0, 1.3 and 0.9 respectively [14].

The two terms on the left hand side of equation (3-10) is the unsteady state and convection of turbulent kinetic energy. On the right hand side of the equation (3-10) are the production of \( k \) due to the mean flow deformation, production of \( k \) by body force, turbulent diffusion and dissipation of \( k \) respectively.

From equation (3-11) it is seen that model \( \varepsilon \) equation reflects the composition of the \( k \) equation except the term with constant \( C_{\varepsilon 4} \) which takes into account the fluid compression on the source of \( \varepsilon \) and does not have its counterpart in the \( k \) equation. This term is effective only for compressible flow because the velocity gradient \( \frac{\partial U_K}{\partial x_K} \) becomes zero in case of incompressible flow.

The turbulent viscosity \( \mu_t \) is defined as follows:

\[ \mu_t = C_{\mu} \rho \frac{k^2}{\varepsilon} \]
In the standard $k - \varepsilon$ turbulence model, a constant value of $C_\mu = 0.09$ is used which was determined from the experimental study where fluid velocity was very low. As described in section 2.2, experimental work has shown that the standard $k - \varepsilon$ model overpredicts the jet spreading rate in case of high speed compressible flow. Heinze [17] modified the value of $C_\mu$ to account for the effect of compressibility which is defined by following equation:

$$C_\mu = 0.07 \exp(-0.4M_g)$$  \hspace{1cm} (3-13)

$$M_g = \frac{|S|}{\alpha}$$  \hspace{1cm} (3-14)

Where $|S|$ is the mean shear rate ($s^{-1}$), $l$ is turbulence length scale ($m$) and $\alpha$ is the speed of sound ($m/s$). This modification of $C_\mu$ accurately predicts the axial velocity distribution of the gas jet at room ambient temperature but still under predicts the axial velocity distribution at high ambient temperatures.

In order to take into account the effect of large temperature gradients, Abdol-Hamid et al.[88] modified the constant $C_\mu$ according to following equations:

$$C_\mu = 0.09C_T$$  \hspace{1cm} (3-15)

$$C_T = \left[1 + \frac{C_1 T_g^m}{1 + C_2 f(M_g)}\right]$$  \hspace{1cm} (3-16)

Where $T_g$ is the function of local total temperature gradient normalized by local turbulence length scale:
Total temperature is also known as the stagnation temperature and is the summation of static and dynamic temperature[11]. The reason for using the local total temperature gradient in their model is that total temperature is not Mach number dependent. Hence this model will not be influenced by internal shocks and flow expansion.

In order to model high speed flow, the turbulence Mach number $M_t$ was included in their modification. The turbulence Mach number and the function of turbulence Mach number was defined as:

$$M_t = \frac{\sqrt{2k}}{\sqrt{a}}$$  \hspace{1cm} 3-18

$$f(M_t) = (M_t^2 - M_{t0}^2)H(M_t - M_{t0})$$  \hspace{1cm} 3-19

Where $a$ is the speed of sound, $H(x)$ is Heaviside function and $M_{t0} = 0.1$. For no compressibility correction, $f(M_t) = 0$. The constants and the coefficients of equation (3.16) are listed in Table 3-2:

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>$m$</th>
<th>$C_1$</th>
<th>$C_2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abdul-Hamid et al.[88]</td>
<td>3</td>
<td>24.33</td>
<td>24.33</td>
</tr>
<tr>
<td>Present Study</td>
<td>0.6</td>
<td>1.2</td>
<td>1</td>
</tr>
</tbody>
</table>
The authors [88] did not derive the functional relationship, constants and coefficients of equation (3.16) analytically. Instead, it was determined by trial and error to fit the total temperature experimental data of Seinor et al. [89] for the supersonic flow. This modification was proposed for a high temperature supersonic jet flowing into the low ambient temperature. In that case, the potential core length of the jet becomes shorter because of the increment in the growth rate of turbulent mixing layer [89]. Nonomura and Fujji [90] also reported that potential core length of the supersonic jet decreases with increasing supersonic jet temperature. The standard $k - \varepsilon$ turbulence model fails to predict the observed increase in the growth rate of turbulent mixing. This modification [88] of $k - \varepsilon$ model was made to increase the turbulent eddy viscosity in the shear layer by increasing the value of $C_\mu$ as a function of total temperature gradient. However, this modification of $C_\mu$ underpredicts the potential core length of the supersonic jet when used for cold supersonic jet exiting into hot atmosphere. This is because the modified model [88] can only increase the growth rate of mixing by increasing the value of $C_\mu$ which in turn makes the potential core length shorter. But, when a cold supersonic jet enters into a hot ambient atmosphere like steelmaking furnace, potential core length should increase due to the reduced ambient density. Hence, a different model is required to accurately simulate the cold supersonic jet emitting into hot ambient temperature.

In this study, the previous model [88] was modified to accurately simulate the cold supersonic jet exiting into a hot atmosphere. Unlike the previous model [88] the value of $C_\mu$ was modified by dividing its standard value 0.09 by the variable $C_T$ to achieve the desired decrease in turbulent viscosity at the shear layer which in turn reduces the growth rate of mixing. The variable $C_T$ was determined by using the similar functional relationship of equation (3.16) used by the previous authors [88]. Following their [88] approach, the coefficients and the constants in equation (3.16) were determined by trial and error for the present study to match accurately the experimental axial velocity distribution data of Sumi et al. [22] at different ambient temperatures. The constants and coefficients for the present study are also listed in Table 3-2. Hence, for the present study equation (3.15) and (3.16) becomes:

\[
C_\mu = \frac{0.09}{C_T}
\]
According to equation (3.20) and (3.21), the value of $C_\mu$ becomes small in the region of high total temperature gradient which in turn reduces the turbulent viscosity. Lower turbulent viscosity leads to lower turbulent shear stress from equation (3.9) and hence lower turbulent mixing leading to increased potential core length. The reduced turbulent shear stress also decreases the turbulence production rate of equation (3.10) which in turn lowers turbulent kinetic energy which further reduces turbulent viscosity through equation (3.12). As a result, growth rate of turbulent mixing region decreases.

The reason for using the ad-hoc modification of the $k - \varepsilon$ turbulence model rather than using other turbulence model is that it is computationally less expensive, robust, and the implementation of the modified term into the solution is comparatively easy. Our objective was to investigate the supersonic jet behaviour at high ambient temperature in order to simulate the supersonic oxygen jet impinging on liquid iron inside the steelmaking furnace. The results obtained with this ad-hoc modification are in good agreement with the experimental data and jet model as will be shown in section 3.4.5.2.

### 3.4 Modelling Of Supersonic Free Jet

In this section, the supersonic jet behaviour at three different ambient temperatures of 285K, 772K and 1002K were numerically investigated and validated against experimental data [22] and a jet model proposed by previous researchers [91]. This CFD model was then used to investigate the supersonic jet behaviour and droplet generation rate at steelmaking temperature of 1800K.

The governing equations and turbulence model used in this simulation have been described in previous section.
3.4.1 Computational Domain

The computational grid used for CFD simulations in the present study is shown in Figure 3-2. The computational grid is an axisymmetric wedge shaped grid with only one cell in circumferential direction. In order to reduce computational time, flow inside the Laval nozzle was not included in the simulation. Flow conditions at the nozzle exit were calculated by using isentropic theory [11]. The exit diameter of the nozzle was 9.2mm and was considered as the inlet to the computational domain. The size of the computational domain was 100 nozzle exit diameters downstream from the nozzle exit and 30 nozzle exit diameters normal to the jet centreline. The mesh had a total of 7760 cells. The grid density was very high at the exit of the nozzle and at the shear layer.

![Figure 3-2 Computational domain with boundary conditions.](image)

3.4.2 Boundary Conditions

All boundary conditions were chosen to match with the experimental study of Sumi et al.[22]. Stagnation pressure boundary condition was used at the inlet of the computational domain (exit of the nozzle). The value of Mach number and temperature were defined at the inlet. At the outlet, static pressure boundary condition was used. For symmetry plane, symmetry boundary condition was used. At the wall, zero heat flux and no-slip boundary condition were used. The values of the boundary conditions are listed in Table 3-3:
Table 3-3 Boundary conditions for the modelling of free supersonic jet.

<table>
<thead>
<tr>
<th></th>
<th>Stagnation pressure</th>
<th>Mach number</th>
<th>Static Temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>497695 Pa</td>
<td>1.72</td>
<td>190 K</td>
</tr>
<tr>
<td>Outlet</td>
<td>Static pressure</td>
<td></td>
<td>100000 Pa</td>
</tr>
<tr>
<td></td>
<td>Temperature</td>
<td>285 K</td>
<td>772 K</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1002 K</td>
<td>1800 K</td>
</tr>
</tbody>
</table>

### 3.4.3 Discretization Schemes

The simulations were carried out using commercial CFD software AVL FIRE 2008.2. This software uses finite volume discretization method to approximate the non-linear partial differential equations by a set of algebraic equations for the variables at some set of discrete locations in space and time. While deriving the algebraic equations for unknown variable \( \Phi \) at certain grid points, some assumptions are made for the calculation of the value of \( \Phi \) at control volume face [84]. These are known as discretization schemes. The way in which the values of variables at cell faces are calculated has a profound effect on the accuracy and convergence of the numerical method. This is particularly important when considering the evaluation of face values for convective transport in case of fully compressible flow. The desirable properties of a discretization scheme are conservativeness, boundedness and transportiveness which are described in detail by Veersteeg and Malalasekera [84]. If the proper discretization scheme is not used for a particular problem or numerical grid is not enough refined, the solution can produce unphysical oscillations and unbounded variable values. In the present study, the first order upwind discretization scheme was used for energy and turbulence equations. The upwind differencing scheme takes into account the flow
direction when determining the value at the cell face: the convected value of $\Phi$ at the cell face is taken to be equal to the value at the upstream node. For momentum and pressure correction equation AVL SMART discretization scheme [92] was used which is a modification of the SMART scheme proposed by Gaskell and Lau [93]. The SMART scheme is a second order accurate TVD (Total variable diminishing) scheme [84]. The accuracy of the AVL-SMART scheme is slightly reduced but the convergence is greatly increased which is more important.

### 3.4.4 Computational Procedure

#### 3.4.4.1 SIMPLE Algorithm

The unsteady, compressible continuity, momentum and energy equations were solved using segregated solver to calculate the pressure, velocity, temperature and density. Every velocity component calculated from the momentum equation for a particular pressure should satisfy the continuity equation. This is done through the use SIMPLE (Semi-Implicit Method for Pressure –Linked Equations) algorithm which was originally put forward by Patankar and Spalding [94] and is essentially a guess and correct procedure for the calculation of pressure on the staggered grid arrangement [84]. In this method, an initial guess for the pressure is made and the velocity field is calculated using discretized momentum equation. The discrete form of the continuity equation is converted into an equation for pressure correction and is solved using the previously calculated velocity. The pressure corrections are then used to update the pressure and velocity fields. The process is repeated until convergence of the velocity and pressure field. In order to advance the solution in time, first order Euler scheme [92] was used. The solution algorithm have been described in detail in the solver manual of AVL FIRE [92]. As the velocity of the flow was very high, time step used in the unsteady calculation was $1 \times 10^{-5}$ s. The simulations were carried out for sufficient time until no further change was observed in the flow field.
3.4.4.2 Under-relaxation

In the iterative solution of the algebraic equations, a change of variable values from the previous iteration ($\Phi^{K-1}$) to the next iteration ($\Phi^K$) is slowed down to ensure convergence of the solution procedure. This process is known as under-relaxation and is done implicitly, as proposed by Patankar [95]:

$$\Phi^K = \Phi^{K-1} + \alpha_\phi(\Phi^{new} - \Phi^{K-1})$$

Where, $\alpha_\phi$ is the under-relaxation factor, $\Phi^{new}$ is the actual variable value calculated from the latest iteration and $\Phi^K$ is the new variable value after under-relaxation. The under-relaxation factor varies from 0 to 1. As the value approach 0, the solution stability increases but it also increase the total processing time. As factor approach 1, the solution stability decreases but may lead to faster solution. Hence, the optimum under-relaxation factor is problem dependent. The under-relaxation factors used in the present study have been presented in Table 3-4:

Table 3-4 Under-relaxation factors

<table>
<thead>
<tr>
<th>Equation</th>
<th>Under-relaxation factors</th>
</tr>
</thead>
<tbody>
<tr>
<td>Momentum</td>
<td>0.6</td>
</tr>
<tr>
<td>Pressure correction</td>
<td>0.1</td>
</tr>
<tr>
<td>Turbulence Kinetic energy</td>
<td>0.4</td>
</tr>
<tr>
<td>Turbulence Dissipation rate</td>
<td>0.4</td>
</tr>
<tr>
<td>Energy</td>
<td>0.8</td>
</tr>
</tbody>
</table>

3.4.4.3 User-Defined Subroutine

The modification of the turbulence model, described in section 3.3.4, was incorporated into the commercial CFD software AVL Fire 2008.2 through user defined subroutine
usevis.f. The code of modified turbulence model was written in the Microsoft Visual Studio 2008 and then compiled using the Intel FORTRAN Compiler 10.1.019 which bridges the Visual Studio with the AVL Fire during simulation. The subroutine was executed at each time step of the simulation.

3.4.5 Results And Discussions

3.4.5.1 Grid Independency Test

In order to study the grid sensitivity of the solution, calculations for 772K ambient temperature were done using four different grid levels: coarse grid (4716 cells), medium grid (7760 cells) and fine grid (10116 cells) and very fine grid (19660). The axial velocity profiles for all grid levels are shown in Figure 3-3. The variation in the axial velocity profiles calculated with medium grid, fine grid and very fine grid is within 5%. Hence, it can be said that the solution is not sensitive to the grid. The solution with very fine grid takes longer time than the solution with medium grid. Therefore, the results obtained with medium grid were used for analysis and discussion in this study.

Figure 3-3 Axial velocity distribution of supersonic oxygen jet at 772K ambient temperature for coarse, medium, fine and very fine grid level.
### 3.4.5.2 Velocity Distribution

The computed velocity using the standard $k - \varepsilon$ model, with compressibility correction of Heinz [17], along the axis of the jet is plotted with experimental data in Figure 3-4.

![Graph showing velocity distribution on center axis with turbulence model including compressibility correction](image)

**Figure 3-4** Velocity distribution on center axis with $k - \varepsilon$ turbulence model including compressibility correction [17].

The graph shows that the potential core length of the jet increases at high ambient temperature. The agreement between CFD and experimental results is very good when the ambient temperature is 285K. But at higher ambient temperature, it fails to accurately predict the velocity distribution. This model underpredicts the potential core length of the jet at high ambient temperatures. The percentage of deviation increases for higher ambient temperatures. The average percentage of deviation of computed velocity from the experimental data is about 13% and 22% for 772K and 1002K ambient temperatures respectively. The ambient temperature inside the steelmaking furnace is about 1800K and the deviation will be much larger at such high temperatures. This is because this turbulence model does not take into account the effect of large temperature
gradients. Figure 3-5 shows that the temperature corrected turbulence model of Abdol-Hamid et al.[88] also underpredicts the potential core length when used for simulating room temperature supersonic jet entering into a 1002K ambient temperature. The reason for the discrepancy has been explained in section 3.3.4.

Figure 3-5 Velocity distribution on centre axis with temperature corrected $k - \varepsilon$ turbulence model of Abdol-Hamid et al.[88].

Figure 3-6 shows the velocity distribution on the centre axis with modified $k - \varepsilon$ model proposed in this study including the effect of high temperature gradient. The modified model predicts more accurately the velocity distribution on centre axis at both high and low ambient temperatures. When the ambient temperature is 772K, the computed value deviates from the experimental value by an average of less than 7%. In case of 1002K ambient temperature the average percentage of deviation of computed velocity from the experimental result is less than 9%. Only CFD result is shown for 1800K ambient temperature because there was no experimental data available at 1800K ambient temperature. Figure 3-6 also shows that although the potential core length of the
Figure 3-6 Velocity distribution on centre axis with the proposed modified $k - \varepsilon$ model.

Figure 3-7 Two dimensional velocity distribution at different ambient temperatures.
supersonic jet is longer at higher ambient temperatures, the relative difference in velocity magnitude at different ambient temperatures reduces beyond \( \frac{x}{d_v} = 30 \). Figure 3-7 shows the two dimensional velocity distribution of the same supersonic jet discharging into different ambient temperatures.

Figure 3-8 shows the dynamic pressure distribution on the centre axis of the jet obtained with the proposed model. As expected, dynamic pressure of the jet is higher at high ambient temperatures which is in contrast with the CFD results of Tago and Higuchi [19] who reported that dynamic pressure of the jet is very little affected by the ambient temperature. The reason for this difference is that they did not use any temperature correction term in their modelling. Dynamic pressure of the jet is a very important characteristic because the higher the dynamic pressure, the bigger the momentum transfers to the liquid bath which results in higher depth of penetration in the liquid bath. This in turn enhances the mixing of oxygen and liquid melt and therefore improves the decarburization rate.

![Figure 3-8 Dynamic pressure distribution on the centre axis of the jet with the proposed model.](image)
The CFD results of the present study underpredicts the experimental dynamic pressure distribution [22] as the distance from the nozzle exit increases. The average percentage of deviation of CFD results from experimental data is about 20%, 17% and 20% for 285K, 772K and 1002K ambient temperatures respectively. This is because dynamic pressure is proportional to the square of velocity. As a result, the errors in the velocity magnitude increased by same proportion. Also dynamic pressure is proportional to density of the gas. The experimental study [22] was performed in an air tight container which may have resulted in an increase in the pressure, as well as density, inside the furnace during the experiment. This increase in density may have resulted in higher dynamic pressure in the experimental study.

The data presented in Figure 3-8 also shows that the relative difference in dynamic pressure of supersonic jets at different ambient temperatures decreases with increasing distance from the nozzle exit. If the distance between the liquid bath and nozzle exit is more than 60 nozzle exit diameter then the effect of high ambient temperatures is negligible because the dynamic pressure of the impinging jet on the liquid bath would be almost same as for all ambient temperatures.

Radial velocity distributions of the supersonic jet at different ambient temperatures at $\frac{x}{d_e} = 5, 22.5$ and 50 are shown in Figure 3-9 to 3-11. With increasing distance from the nozzle exit, the jet spreads and its axial velocity decreases. But at high ambient temperatures the axial velocity of the jet decreases at a slower rate compared to lower ambient temperature. This is because the mass of entrained fluid from the surrounding medium is low due to the low ambient density. Figure 3-10 shows that at $\frac{x}{d_e} = 22.5$, the supersonic jet at 1800K ambient temperature still maintains potential core length where the axial velocity of the jet at room ambient temperature becomes less than half of jet exit velocity. Figure 3-9 to 3-11 also shows that although the high ambient temperature increases the potential core length of the jet, the width of the jet is little affected by the ambient temperature after the potential core region.
Figure 3-9 Radial distribution of supersonic jet at different ambient temperatures at $\frac{x}{d_e} = 5$.

Figure 3-10 Radial distribution of supersonic jet at different ambient temperatures at $\frac{x}{d_e} = 22.5$. 
Figure 3-11 Radial distribution of supersonic jet at different ambient temperatures at $\frac{x}{d_e} = 50$.

Figure 3-12 shows the spreading rate of the supersonic jet at different ambient temperatures. The spreading rate is defined as [96],

$$S_p = \frac{r_{1/2}}{x-x_0}$$  \hspace{1cm} 3-22

Where $S_p$ is the spreading rate, $r_{1/2}$ is the width of the half value of axial velocity, $x_0$ is the potential core length of the jet. Figure 3-12 shows that near the nozzle exit the spreading rate of the jet is only 0.025 ($1.54^0$ spreading angle) at all ambient temperatures. Then after a certain distance from the nozzle exit, the spreading rate of the jet becomes around 0.1 ($6.2^0$ spreading angle) and is very little affected by ambient temperature. This distance is the potential core length of the jet and is longer at high ambient temperature as seen from Figure 3-12. This value of spreading rate after the
potential core length of the jet is in close agreement with the experimental study [22] where spreading angle of 6° was obtained.

Figure 3-12 Spreading rate of supersonic jet at different ambient temperatures.

3.4.5.3 Temperature Distribution

The calculated temperature distribution on the centre axis of the jet is shown in Figure 3-13 for 285K, 772K, 1002K and 1800K ambient temperatures. The figure shows that the temperature of the gas jet increases gradually after the discharge from nozzle exit and tends to reach the ambient temperature. The computed jet temperature is in very good agreement with experimental data for 285K and 772K ambient temperature and in reasonable agreement for 1002K ambient temperature. The computed values differ from the experimental results by less than 2% for 285K and 772K ambient temperature. For 1002K ambient temperature, the average percentage of deviation is about 7% except at the point $\frac{x}{d_e} = 21$. The reason for this difference is that the numerical model slightly underpredicts the heat transfer through the turbulent shear layer from the ambient to the
jet for 1002K ambient temperature. In the present model, turbulent transport of heat has been calculated by equation (3.6) where a constant value of $Pr_t = 0.5$ [86] is used in all cases. But value of $Pr_t$ should be different for different temperature gradient. Because, if the temperature gradient is large, the turbulent heat transfer rate will be high. Decreasing the $Pr_t$, enhances the heat diffusion capacity in the flow. The manner in which $Pr_t$ varies with different temperature gradient is the subject of further research.

![Figure 3-13 Temperature distribution on centre axis with the proposed model.](image)

**3.4.5.4 Variation Of Constant $C_\mu$ With Temperature**

As discussed earlier, the effect of the temperature corrected $k-\varepsilon$ turbulence model, proposed in the present study, over the standard $k-\varepsilon$ model is to decrease the growth rate of turbulent mixing region and hence increase the potential core length of the jet. This is done by decreasing the turbulent viscosity closure coefficient $C_\mu$ in the shear mixing layer depending on the temperature gradient. The action of the modified $k-\varepsilon$ turbulence model at different ambient temperature is shown in Figure 3-14 to 3-17. The
figures show that the value of $C_\mu$ in the shear mixing layer decreases with increasing temperature gradient which in turn results in lower turbulent viscosity. The lowest value of $C_\mu$ is around 0.064, 0.041, 0.036 and 0.026 at 285K, 772K, 1002K and 1800K respectively. The decreased turbulent viscosity eventually reduces the mixing of the jet with the surrounding and increases the potential core length.

Figure 3-14 Values of coefficient $C_\mu$ at 285K ambient temperature using present model.

Figure 3-15 Values of coefficient $C_\mu$ at 772K ambient temperature using present model.
Figure 3-16 Values of coefficient $C_\mu$ at 1002K ambient temperature using present model.

Figure 3-17 Values of coefficient $C_\mu$ at 1800K ambient temperature using present model.

Figure 3-18 to Figure 3-20 show the values of the coefficient $C_\mu$ at different ambient temperature using the compressible correction of Heinz [17]. The lowest value of $C_\mu$ at 285K ambient temperature has been calculated as 0.066 which is in good agreement with the present study. But at high ambient temperatures the compressibility correction model of Heinz predicts almost similar values of $C_\mu$ and hence underpredicts the potential core length.
Figure 3-18 Values of coefficient $C_\mu$ at 285 K ambient temperature using Heinz model [17].

Figure 3-19 Values of coefficient $C_\mu$ at 772 K ambient temperature using Heinz model [17].

Figure 3-20 Values of coefficient $C_\mu$ at 1002K ambient temperature using Heinz model [17].
Abdul-Hamid et al.[88] reported that the maximum value of $C_{\mu}$ is reached 0.27 (three times the normal value 0.09) when a hot jet of 1116K total temperature is discharged into room ambient temperature. In the present study, it was found that the minimum value of $C_{\mu}$ becomes 0.036 (just over one third of the normal value 0.09) when a room temperature jet discharges into a hot (1002K) atmosphere. Hence, from the present study and the previous study of Abdul-Hamid et al.[88], it can be said that the value of $C_{\mu}$ should be different when there is a large temperature gradient between the jet and the ambient where the jet is being discharged.

### 3.4.5.5 Comparison With Jet Model

The numerical results obtained with the present model were compared with the empirical jet model of Ito and Muchi [91] given in equation (3.23):

\[
\frac{1}{2 \ln(1-U_m)} = \alpha \frac{\rho_a x}{\rho_g d_e} - \beta 
\]

\[U_m = \frac{U}{U_0}\]

Where $U_0$ is the velocity at the nozzle exit, $\rho_g$ is the density of the gas at nozzle exit and $\rho_a$ is the ambient density. Sumi et al.[22] calculated the values of constants $\alpha =$0.0841 and $\beta =$0.06035 from their experimental study. Figure 3-21 shows the velocity ratio obtained from the present CFD model as a function of $\sqrt[2/3]{\frac{\rho_a x}{\rho_g d_e}}$ for different ambient temperatures. Velocity ratio obtained from the equation (3.23) and from the experimental study is also shown in the same figure. The CFD model agrees well with the empirical jet model at both high and low ambient temperatures with a maximum of 10% deviation. No experimental data was available for the jet behaviour at 1800K ambient temperature. But it is seen that the velocity ratio at 1800K ambient temperature is also in good agreement with the empirical jet model. This shows that the present CFD model should be useful in predicting the jet behaviour in high ambient temperatures.
Figure 3-21 Comparison of velocity ratio with equation (3.23) as a function of $\sqrt{\frac{\rho_a}{\rho_g}} \sqrt{\frac{x}{d_e}}$

### 3.4.5.6 Potential Core Length

From the preceding sections it is already known that the potential core length of the supersonic jet increases at high ambient temperatures. Sumi et al. [22] calculated potential core length of the jet at different ambient temperatures from equation (3.23) by taking $U_m=1$. If $U_m=1$ then the left hand side of equation (3.23) becomes zero and after some manipulation the equation (3.23) becomes:

$$\frac{x}{d_e} = \frac{\beta}{\alpha \sqrt{\frac{\mu_a}{\rho_g}}}$$

Using the above mentioned values of the constants and appropriate density, potential core length of the jet was determined for different ambient temperatures. Allemand et
al.[12] also proposed an empirical equation for calculating the potential core length of the jet at different ambient temperature which is:

\[ \frac{x}{d_e} = \sqrt{\frac{\rho_g}{\rho_a}} \left( 4.2 + 1.1 \left( Ma^2 + 1 - \frac{T_e}{T_a} \right) \right) \]

Equation (3.26) is a modification of the original equation proposed by Lau et al.[97] which is suitable for calculating potential core length only at room ambient temperature. Figure 3-22 shows the potential core length of the jet at different ambient temperatures calculated by equation (3.25), (3.26) and from the present CFD model. There is a high correlation between potential core length calculated by equation (3.25) and the results obtained by present CFD model. At steelmaking temperatures (1800K), the potential core length of the supersonic jet becomes 2.5 times of the potential core length at room temperature.

Figure 3-22 Coherent length of supersonic jet at different ambient temperatures.
temperature (285K). The potential core length calculated by equation (3.26) is higher than those obtained by equation (3.25) and present CFD model at all ambient temperature. No experimental data of the potential core length at high ambient temperature is available in the literature. Hence, it is unknown which empirical equation is correct.

3.4.5.7 Droplet Generation

In order to quantify the influence of high ambient temperature on the droplet generation rate, Blowing number \( (N_B) \) was calculated for the supersonic jet at different ambient temperatures. The Blowing number is a dimensionless number proposed by Subagyo et al.[7] and was presented in equation (2.32). Figure 3-23 shows the variation of Blowing number with distance from the nozzle exit (it can be assumed as the distance between nozzle exit and liquid bath) at different ambient temperatures. The surface tension of the liquid melt was taken as 1.9 N/m assuming that the liquid melt is Fe [98]. The variation of surface tension with temperature and composition was not considered because the transfer of jet momentum on liquid surface is the dominant factor for droplet generation compared to the changes in liquid properties [99]. The density of the liquid melt was taken as 7030 Kgm\(^{-3}\).

Figure 3-23 shows that the Blowing number is greater for the gas jet at higher ambient temperature. When the distance between nozzle exit and liquid bath is around 40\(d_e\), \(N_B=6\) for 1800K ambient temperature compared to \(N_B=3\) for 285K ambient temperature. Figure 2-21 shows the droplet generation rate at different Blowing number in case of top jetting [7]. The figure shows that the droplet generation rate at \(N_B=6\) increases significantly compared with the droplet generation rate at \(N_B=3\). Hence, the effect of ambient temperature on the generation of droplets needs special consideration to optimize the process.
3.4.6 Conclusions

The behaviour of supersonic oxygen jet in a high temperature field is investigated by CFD simulation. The results obtained from this study are as follows:

- The standard $k - \varepsilon$ turbulence model with compressibility correction under predicts the potential core length of the supersonic jet at high ambient temperatures.

- A modification of the standard $k - \varepsilon$ turbulence model has been proposed by adjusting the value of $C_{\mu}$ depending on temperature gradient. The validity of this model has been confirmed by comparing the velocity and temperature profile obtained by CFD model with the available experimental data.

- The potential core length of supersonic oxygen jet at steelmaking temperature (1800K) is 2.5 times than that of room ambient temperature. The dynamic pressure of the jet, which characterizes the momentum transfer, is higher at high ambient temperature but with increasing distance from the nozzle exit the
relative difference between the dynamic pressure value at different ambient
temperature decreases and the effect of ambient temperature becomes less.

- The spreading rate of the jet is not affected by the ambient temperature after the
  potential core length of the supersonic jet.

- The ambient temperature has an influence on droplet generation rate. The
  Blowing number of the supersonic jet at high ambient temperature is higher than
  the room ambient temperature. As a result, droplet generation rate is higher.

However, this model is not suitable for the simulation of a high temperature supersonic
jet flowing into low ambient temperature. Although the trial and error approach has
been used in the present study to modify the turbulence model and a more rigorous
approach is desirable, the present study will give useful information for developing
theoretical turbulence model including the effect of temperature gradient and contribute
to the understanding of jet behaviour at high ambient temperatures.
3.5 Modelling Of Supersonic Coherent Jet

The governing equations, described in section 3.2 were used for the simulation of supersonic coherent jet except that the energy equation contains an extra source term:

\[
\frac{\partial \rho H}{\partial t} + \frac{\partial (\rho H u_i)}{\partial x_i} = - \frac{\partial}{\partial x_i} \left( q_i + c_p \rho u_i T^i \right) + \frac{\partial}{\partial x_i} \left( \tau_{ij} U_j - \rho u_i u_j U_j \right) + \frac{\partial p}{\partial t} + S_c - S_E \quad 3-27
\]

Where \( S_c \) is the source of heat energy from the combustion flame surrounding the main supersonic jet in case of coherent flow and \( S_E \) is the radiation heat transfer. In order to model the combustion flame, an extra species transport equation was solved. For the turbulence modelling, temperature corrected turbulence model, discussed in section 3.3.4, was used. The following assumption was made for the simulation of coherent jet:

- Single step complete combustion between the fuel and oxidant.

This assumption is justified in the later sections of this chapter.

3.5.1 Combustion Modelling

The fuel and oxidizing agent used in the present study were CH\(_4\) and O\(_2\). N\(_2\) and O\(_2\) were used as the central supersonic jet. As stated earlier, single step complete combustion reaction was assumed in the current study and the products of combustion were CO\(_2\) and H\(_2\)O. In practice, however, at high temperatures dissociation of CO\(_2\) and H\(_2\)O occurs which results minor species like CO, H\(_2\), and OH in the products of combustion along with main reaction products CO\(_2\) and H\(_2\)O [84]. Dissociation reactions are endothermic: therefore the actual flame temperature will be lower than the calculated flame temperature based on complete combustion reaction [84]. But this assumption of one step complete combustion reaction made the calculation simple and also predicted the axial velocity distribution with good accuracy as will be shown in section 3.5.6.2. The equation of combustion reaction is given by [84],
\[ \text{CH}_4 + 2\text{O}_2 = \text{CO}_2 + 2\text{H}_2\text{O} \]  

16Kg  64Kg  44Kg  36Kg

The mass fraction of the species involved in the reaction are determined by solving separate scalar transport equation for each species which can be written in conservation form as,

\[ \frac{\partial \rho Y_i}{\partial t} + \frac{\partial (\rho Y_i u_j)}{\partial x_j} = -\frac{\partial}{\partial x_j}\left((\rho D_i + \frac{\mu_t}{Sc_t})\frac{\partial Y_i}{\partial x_j}\right) + S_i \]  

Where \( Y_i \) is the mass fraction, \( D_i \) is the laminar diffusion coefficient (\( m^2/s \)) and \( S_i \) is the source term of species \( i \). In the present study, a constant diffusion coefficient of the gas mixture was used for all species i.e. \( D_i = D \) for \( i = 1, 2, 3 \ldots \ldots \ldots N \) where \( N \) is the number of species. The gas mixture diffusion coefficient was calculated assuming the laminar Schmidt number, \( Sc_l = 0.7 \). As the flow is highly compressible, the laminar diffusion coefficient will have negligible effect on the diffusion of the species. It is the turbulent diffusion coefficient which affects the diffusion of different species in the flow field. The turbulent diffusion coefficient was determined by dividing the turbulent viscosity \( \mu_t \) of the gas mixtures by the turbulent Schmidt number, \( Sc_t = 0.9 \). Hence, it is seen that the total diffusion coefficient was same for all species involved in the reaction. The source term of the species transport equation is the rate of mass production/reduction of that particular species. When species transport equation for the fuel (\( \text{CH}_4 \)) was solved, the rate of fuel consumption was determined by solving the Eddy-Break-up combustion model [100]:

\[ S_{fu} = \rho \frac{\varepsilon}{k} \min \left[ AY_{fu}, A \frac{Y_{ox}}{s}, B \frac{Y_{pr}}{s+1} \right] \]
Where $S_{fu}$ is the volumetric rate of fuel consumption($kgm^{-3}s^{-1}$). $A$ and $B$ are the model constants and $s$ is the stoichiometric ratio. $Y_{fu}$, $Y_{ox}$ and $Y_{pr}$ are the mass fraction of the fuel, oxygen and product of combustion respectively. The eddy break up model makes good predictions and is fairly straightforward to implement in CFD calculations [84]. In the Eddy Break-up model, the rate of fuel consumption is specified as a function of local flow and thermodynamic properties. According to this model, the rate of combustion is determined by the rate of intermixing on a molecular scale of eddies containing reactants and those containing hot products, in other words by the rate of dissipation of this eddies. This model calculates the individual dissipation rates of fuel, oxygen and products and the actual consumption rate is equal to the slowest of the three dissipation rates as seen from the equation (3.30) [84]. The first two terms inside the bracket of equation (3.30) simply determine whether fuel or oxygen is present in limiting quantity, and the third term ensures that the flame does not spread in the absence of hot products. In the present study, the values of $A=4$ and $B=0.5$ were used based on a previous study [100]. From the combustion reaction, it is seen that the stoichiometric ratio of fuel and oxidizer is $s=4$ which means that for complete combustion of 1kg CH$_4$, 4kg O$_2$ is required. Hence, the rate of O$_2$ consumption is taken as 4 times the rate of fuel consumption. The volumetric rate of fuel consumption $S_{fu}$ was calculated in each cell by using equation (3.30) which is then multiplied by the lower heating value of CH$_4$ and added as a source term to the energy equation to calculate the temperature. Hence the source term $S_c$ ($J/m^3s$) of the energy equation (3.27) can be written as:

$$S_c = Q_H \times S_{fu}$$

(3-31)

The lower heating value of the reaction presented in equation (3.28) is $Q_H = 50000000$ J/kg [84]. Heating value of a species is the amount of heat released during the combustion of the specified amount of it. If H$_2$O is present in vapour form in the product of combustion then it is known as lower heating value. In the present study, lower heating value was used because H$_2$O in the product is in vapour phase.
3.5.2 Radiation Modelling

Radiation heat transfer from a system becomes important when the temperature exceeds 1500K [84]. Here the flame temperature of the combustion is around 3500K. So the radiation heat transfer needs to be considered. The modelling of the radiation was performed using the well-known Stefan-Boltzmann formula:

\[ E = \varepsilon \sigma_s A(T_1^4 - T_2^4) \]  

Where \( E \) is the radiation heat transfer per unit time (W), \( \varepsilon \) is the gas emissivity, \( \sigma_s = 5.6703 \times 10^{-8} \text{W/m}^2\text{K}^{-4} \) is the Stefan-Boltzmann constant, \( A \) is the area of emitting body, \( T_1 \) and \( T_2 \) are the temperature of source and sink respectively. The emissivity of a medium depends on local fluid properties [84]. The normal atmospheric air is transparent and thus does not participate in radiation heat exchange. Products of combustion, on the other hand, contain a high concentration of CO\(_2\) and H\(_2\)O which are both strong absorbers and emitters. The weighted sum of grey gases model (WSGGM) [101] is normally used for defining the temperature and species concentration dependent emissivity of the medium. For different gas concentration and temperature, the emissivity of gases was found to vary from 0.3 to 0.5 using the WSGGM model. In the present study, a constant value of the gas emissivity \( \varepsilon = 0.5 \) was used for simplicity. The radiation energy \( E \) was calculated in each cell using equation (3.32) and then subtracted from the energy equation.

3.5.3 Computational Domain

The schematic diagram of the computational domain with boundary conditions, used in the present CFD simulation, is shown in Figure 3-24. The computational domain is axisymmetric and wedge shaped with only one cell in circumferential direction. In order to reduce computational time, flow inside the Laval nozzle was not included in the
simulation. Flow conditions at the nozzle exit were calculated by using isentropic theory [11]. The exit diameter of the nozzle was 14.7mm and was considered as one of the inlets to the computational domain. The computational domain was 105 nozzle exit diameters downstream from the nozzle exit and 20 nozzle exit diameters normal to the jet centreline. In the experimental study, CH\textsubscript{4} and O\textsubscript{2} were injected through the holes arranged in two concentric rings surrounding the main convergent-divergent nozzle as shown in Figure 2-10. The inner ring of holes was used for CH\textsubscript{4} gas and outer ring of holes was used for shrouding oxygen supply. The diameters of the inner and outer rings were 0.0413m and 0.057m respectively. The diameters of the holes were 0.00287m and 0.00408m for CH\textsubscript{4} and O\textsubscript{2} respectively. In the present study, the configuration of the nozzle, injecting CH\textsubscript{4} and shrouding O\textsubscript{2}, was assumed annular which is different from three dimensional real nozzles. However, injecting areas were adjusted to maintain the same velocity and flow rates for CH\textsubscript{4} and shrouding O\textsubscript{2} as was used in the experimental study [25]. This assumption made it possible to solve the problem in two dimensions.

3.5.4 Boundary Conditions

All boundary conditions were chosen to match with the experimental study of Anderson et al.[25] Stagnation pressure boundary condition was used at the main supersonic jet
inlet of the computational domain (exit of the convergent-divergent nozzle). The value of Mach number and temperature were defined at the supersonic jet inlet. For the CH$_4$ and shrouding O$_2$ inlet, mass flow rate boundary condition was used. The original mass flow rates were divided by 360 because the included angle of the two dimensional computational domain is one degree. At the outlet, a static pressure boundary condition was used. For symmetry plane, symmetry boundary condition was used. At the solid wall, no-slip boundary condition was imposed. The values of the boundary conditions are listed in Table 3-5 and 3-6.

Table 3-5 Boundary conditions for coherent oxygen jet simulation.

<table>
<thead>
<tr>
<th>Name of Boundary</th>
<th>Types of Boundary conditions</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Supersonic jet inlet</td>
<td>Stagnation Pressure</td>
<td>914468Pa</td>
</tr>
<tr>
<td></td>
<td>Mach Number</td>
<td>2.1</td>
</tr>
<tr>
<td></td>
<td>Total Temperature</td>
<td>298K</td>
</tr>
<tr>
<td></td>
<td>Mass fractions</td>
<td>O$_2$=100%</td>
</tr>
<tr>
<td>Fuel inlet</td>
<td>Mass Flow Rate</td>
<td>$1.833 \times 10^{-5}$Kg/s</td>
</tr>
<tr>
<td></td>
<td>Mass fractions</td>
<td>CH$_4$=100%</td>
</tr>
<tr>
<td>Shrouding Oxygen inlet</td>
<td>Mass Flow Rate</td>
<td>$3.488 \times 10^{-5}$Kg/s</td>
</tr>
<tr>
<td></td>
<td>Mass fractions</td>
<td>O$_2$=100%</td>
</tr>
<tr>
<td>Outlet</td>
<td>Static Pressure</td>
<td>100000Pa</td>
</tr>
<tr>
<td></td>
<td>Mass fractions</td>
<td>O$_2$=23%, N$_2$=77%</td>
</tr>
<tr>
<td>Wall</td>
<td>No-slip</td>
<td>298K</td>
</tr>
</tbody>
</table>

The only difference between the boundary conditions of the coherent oxygen and nitrogen jet is the mass fractions at supersonic jet inlet. When nitrogen was used as central supersonic jet, only the supersonic jet inlet mass fraction was changed from 100% oxygen to 100% nitrogen and rest of the boundary conditions remain unchanged. Although it is known that the different gases lead to different static pressures and temperatures for similar stagnation conditions, we have used similar stagnation conditions for both nitrogen and oxygen jet to match with the experimental study.
Table 3-6 Boundary conditions for coherent nitrogen jet simulation.

<table>
<thead>
<tr>
<th>Name of Boundary</th>
<th>Types of Boundary conditions</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Supersonic jet inlet</td>
<td>Stagnation Pressure</td>
<td>914468Pa</td>
</tr>
<tr>
<td></td>
<td>Mach Number</td>
<td>2.1</td>
</tr>
<tr>
<td></td>
<td>Total Temperature</td>
<td>298K</td>
</tr>
<tr>
<td></td>
<td>Mass fractions</td>
<td>N₂=100%</td>
</tr>
<tr>
<td>Fuel inlet</td>
<td>Mass Flow Rate</td>
<td>1.833×10⁻⁵Kg/s</td>
</tr>
<tr>
<td></td>
<td>Mass fractions</td>
<td>CH₄=100%</td>
</tr>
<tr>
<td>Shrouding Oxygen inlet</td>
<td>Mass Flow Rate</td>
<td>3.488×10⁻⁵Kg/s</td>
</tr>
<tr>
<td></td>
<td>Mass fractions</td>
<td>O₂=100%</td>
</tr>
<tr>
<td>Outlet</td>
<td>Static Pressure</td>
<td>100000Pa</td>
</tr>
<tr>
<td></td>
<td>Mass fractions</td>
<td>O₂=23%, N₂=77%</td>
</tr>
<tr>
<td>Wall</td>
<td>No-slip</td>
<td>298K</td>
</tr>
</tbody>
</table>

3.5.5 Computational Procedure

The computational procedure followed for the simulation of the coherent jet is exactly similar to that described in section 3.4.4 except the user defined subroutine which is described in the following section.

3.5.5.1 User Defined Subroutine

As discussed earlier, the coherent jet is created by enveloping the central supersonic jet with a combustion flame. But the AVL Fire software does not allow combustion if supersonic flow is selected under compressibility tab. Hence, the modelling of the shrouding combustion flame was carried out using a group of user defined subroutines which are presented in Table 3-7 along with their functions:
Table 3-7 List of user defined subroutines used in the simulation.

<table>
<thead>
<tr>
<th>Subroutines</th>
<th>Functions</th>
</tr>
</thead>
<tbody>
<tr>
<td>usedef.f</td>
<td>Defines the number of scalar</td>
</tr>
<tr>
<td>useini.f</td>
<td>Variable initialization</td>
</tr>
<tr>
<td>usebnd.f</td>
<td>Boundary condition update</td>
</tr>
<tr>
<td>usepsc.f</td>
<td>Updates source terms of scalar equations</td>
</tr>
<tr>
<td>useent.f</td>
<td>Enthalpy update</td>
</tr>
<tr>
<td>usecps.f</td>
<td>Specific heat update</td>
</tr>
<tr>
<td>usevis.f</td>
<td>Turbulent viscosity update</td>
</tr>
</tbody>
</table>

First of all, the subroutine usedef.f was used to define the number of scalar equations solved in the simulation. In the present study, five different scalar equations were solved for CH₄, O₂, CO₂, H₂O and N₂. Then useini.f was used to initialize the value of different species in the domain at the start of the simulation. After that usebnd.f subroutine was used to define the boundary conditions for the species used in the simulation. Once the calculation started, the usepsc.f subroutine was used to update the source terms of the scalar equations of different species following the procedure described in section 3.5.1. The useent.f subroutine updated the enthalpy at each time step according to the exothermic combustion reaction and heat transferred by radiation. The specific heat values of the different species changes with temperature which was taken into account in the present study through usecps.f subroutine. Finally, the usevis.f subroutine was used to update the turbulent viscosity according to the modified turbulence model described in section 3.3.4.
3.5.6 Results and Discussions

3.5.6.1 Grid Sensitivity Test

In order to study the grid sensitivity of the solution, calculations for the supersonic coherent oxygen jet were done using three different grid levels: coarse grid (20100 cells), medium grid (28000 cells) and fine grid (39000 cells). The axial velocity profile for all grid levels is shown in Figure 3-25:

Figure 3-25 Predicted axial velocity distributions at the jet centreline of shrouded oxygen jet using coarse, medium and fine grid levels.

The average percentage of variation of the axial velocity profile calculated with the coarse and medium grid level was less than 3% with a maximum deviation of 6% between the region of $\frac{x}{d_e} = 40$ and 60. Average percentage of variation was calculated by averaging the differences at several locations in axial direction. Between medium and fine grid level, the variation is negligible (less than 1%). Hence it can be said that the solution is not sensitive to the grid. The computational time required for the fine
grid level was approximately twice than that for medium grid level. Hence, the results obtained with the medium grid were used for analysis and discussion in the present study.

3.5.6.2 Velocity Distribution

Figure 3-26 shows the predicted velocity distribution of the supersonic oxygen jet with and without shrouding flame at room ambient temperature. The supersonic jet without the shrouding flame will be addressed as conventional jet from here after. For both cases, the centreline jet velocity shows repeated fluctuations just after the exit from the Laval nozzle. This occurs due to the incorrect expansion of the supersonic jet. The jet is mildly under-expanded because the ratio of nozzle exit pressure to the ambient pressure in the present study is around 1.18 [102].

![Figure 3-26 Axial velocity distributions at the jet centreline of supersonic oxygen jet with and without shrouding flame.](image-url)
The potential core length of the supersonic oxygen jet with shrouding flame is found to be more than 4 times larger than that without shrouding flame. The velocity of the conventional oxygen jet is predicted to decrease gradually after 10 nozzle exit diameter from the nozzle exit plane. With shrouding combustion flame, the oxygen jet remains coherent up to 42 nozzle exit diameters before the velocity starts to decrease. The shrouding flow injection and the subsequent combustion affect the compression and expansion wave structure within the main jet. The reason for the longer coherent length of the shrouded jet is the reduction in the growth rate of turbulent mixing layer due to the combustion flame which has been described in section 3.5.6.3. Papamoschou and Roshko [13] showed that the growth rate of turbulent mixing layer decreases when the ratio of surrounding ambient density to the jet density decreases. The combustion flame creates a low density region surrounding the main supersonic jet as shown in Figure 3-27 and thus reduces the growth rate of turbulent mixing region. As a result, the shrouded jet spreads slowly compared to the conventional jet.

![Predicted density distribution in case of shrouded oxygen jet.](image)

Figure 3-27 Predicted density distribution in case of shrouded oxygen jet.
The CFD results are in good agreement with the experimental results [25] in case of conventional jet. For the shrouded jet, the CFD model overpredicts the axial velocity by 6% in the coherent region. This may be due to the assumption of using annular rings instead of discrete holes for the CH₄ and shrouding O₂ inlets. Injection from the discrete holes increases the mixing and as a result of that the experimental axial velocity of the jet is lower than the calculated velocity up to a certain distance after exiting from the Laval nozzle. The calculated jet velocity shows relatively quick diffusion compared to the experimental velocity after the coherent region. At $\frac{x}{d_e}$=50 and 60, the difference between CFD and experimental result is about 30%. After 70 nozzle exit diameters CFD result shows good agreement with experimental velocity. The reason for the quick diffusion after coherent region may be due to the use of single total diffusion coefficient for all different species involved in the combustion. Another reason may be the assumption of one step combustion reaction in the combustion modelling. In the real situation, this reaction has a number of steps. CO₂ dissociates into CO and O₂ at high temperature of around 1500K [84]. Oxygen from the periphery of the jet reacts with the CO which further reduces the growth rate of turbulent mixing layer.

The difference between the numerical and experimental study may also have been resulted from the uncertainties involved in the numerical procedure. The possible sources of uncertainties are:

- The turbulence model used in the simulation. The modified $k - \varepsilon$ model used here (discussed in section 3.3.4) was developed for simulating the supersonic gas jet behaviour at steelmaking temperature where no combustion was involved. This modified $k - \varepsilon$ model may result some uncertainties in flow velocity and flame temperature predictions when simulating the turbulent combusting flow. Jones and Whitelaw [103] reported some discrepancies in the measured and predicted velocity and temperature contours in their calculation of turbulent combusting flow using the standard $k - \varepsilon$ model.
- Discretization of partial differential equations. Effort was made to overcome this error by using fine grids.
The differencing schemes used for solving the RANS equations. These differencing schemes introduce numerical diffusion error in the solution. As discussed earlier, higher order schemes (AVL SMART) were used in momentum and continuity equation to minimize the numerical diffusion errors.

The individual analysis of numerical error generated by $k - \varepsilon$ turbulence model, differencing schemes, combustion model and discretization procedure in the solution is beyond the scope of this study. But it can be said that the total numerical uncertainties in the solution is not more than the difference between experimental and numerical results which is 6% in the coherent region and 30% in the region between $x/a_e = 50$ and 60.

Figure 3-26 also shows the predicted axial velocity distribution of the supersonic coherent oxygen jet at steelmaking conditions. In this study, we have considered only air at 1700K temperature as the steelmaking condition. But in reality the furnace environment consists of CO, CO$_2$, O$_2$, H$_2$, N$_2$ and some other minor species. The potential core length of the coherent oxygen jet is around 58 nozzle exit diameters at steelmaking condition which is approximately 1.4 times larger than that of the coherent oxygen jet at room ambient temperature. This occurs because after the combustion flame, the density of the gases surrounding the supersonic jet approaches to the ambient density which is much lower in case of steelmaking conditions. Hence, the jet spreads more slowly at steelmaking conditions because of the low density ratio. No experimental data is available in the literature for the coherent jet at steelmaking temperatures because it is very difficult to perform experimental studies at such high temperatures.

Figure 3-28 shows the predicted velocity distribution of the supersonic nitrogen jet with and without shrouding flame at room ambient temperature. The potential core length of the shrouded nitrogen jet is over 3 times longer than the conventional nitrogen jet. The shrouded nitrogen jet remains coherent up to 32 nozzle exit diameters compared to 10 nozzle exit diameters for conventional jet. The CFD results for the shrouded nitrogen jet are in good agreement with the experimental data with an average of 6% deviation only.

From Figure 3-26 and 3-28, it is evident that the increase in potential core length by the use of shrouding flame is higher for supersonic oxygen jet compared to the supersonic
nitrogen jet. The explanation behind this observation has been described in section 3.5.6.3.

![Figure 3-28 Axial velocity distributions at the jet centreline of supersonic nitrogen jet with and without shrouding flame.](image1)

Figure 3-28 Axial velocity distributions at the jet centreline of supersonic nitrogen jet with and without shrouding flame.

![Figure 3-29 Half jet width of the supersonic oxygen and nitrogen jets with and without shrouding flame.](image2)

Figure 3-29 Half jet width of the supersonic oxygen and nitrogen jets with and without shrouding flame.
Figure 3-29 shows the dimensionless half jet width of the supersonic oxygen and nitrogen jet with and without shrouding combustion flame. “Half jet width” refers to the radial distance from the jet centreline where the velocity of the jet becomes half of the axial velocity. The figure shows that the half width of the jet is similar for conventional nitrogen and oxygen jet. The jet width increases slowly up to $\frac{x}{d_e} = 10$ which is the coherent region and then starts to increases at a higher rate. For shrouded oxygen jet, the jet width increases just after the exit from the nozzle up to $\frac{x}{d_e} = 1$ and then it increases very slowly up to 42 nozzle exit diameters and after that it starts to increase at a higher rate. The reason for rapid increase in jet width just after the exit from the nozzle is the additional combustion at the periphery of the central $O_2$ jet. Due to the combustion, the density of the gases is low at this region which in turn accelerates the gas mixtures at the jet periphery equal to the supersonic jet velocity and results in the increase of jet width just after the exit from the Laval nozzle. In case of the supersonic shrouded nitrogen jet, the jet width increases slowly up to $\frac{x}{d_e} = 32$ and then increases at a higher rate. This rate of increase of jet width can also be defined as the jet spreading rate of the jet [96]:

$$S_p = \frac{r_{1/2}}{x=x_0}$$

3-33

Where $S_p$ is the jet spreading rate, $r_{1/2}$ is the width of the half value of jet axial velocity and $x_0$ is the potential core length of the jet. Figure 3-29 shows that spreading of the jet is restrained by the use of shrouding combustion flame. In other words, the shrouding combustion flame reduces the entrainment of the ambient fluid into the central supersonic jet dramatically. The figure also shows that all of the four different jets spread at a constant rate after the potential core length of the jet. The spreading rate is 0.107 for all four cases which is in excellent agreement with the theoretical spreading rate of 0.1 for a free turbulent jet [96]. This is because after the potential core region the flow becomes fully turbulent and acts as a free turbulent jet for all cases.
3.5.6.3 Temperature Distribution

Figure 3-30 shows the predicted static axial temperature distribution for both the supersonic oxygen and nitrogen jet with shrouding combustion flame. The temperature of the supersonic jets shows some fluctuation after exiting from the Laval nozzle, increases rapidly from flame end position to a maximum value and then decreases slowly to ambient temperature. The reason for the difference in static temperature of O\textsubscript{2} and N\textsubscript{2} jet after exiting from Laval nozzle is the use of similar stagnation temperature for both jets. The different gases lead to different static temperatures for the same stagnation temperature. Sumi et al.[22] also observed the similar axial static temperature distribution in their experimental study. This is most likely because at the end of the coherent length the central jet mixes with surrounding hot atmosphere created by the combustion flame and the temperature of the jet increases. After that heat transfer occurs from the jet to the ambient fluid and the temperature of the jet slowly approaches to the ambient condition. The numerical study of Jeong et al.[104] did not predict this type of behaviour.

![Figure 3-30 Predicted axial static temperature distributions at the jet centreline of shrouded oxygen and nitrogen jet.](image)

Figure 3-30 Predicted axial static temperature distributions at the jet centreline of shrouded oxygen and nitrogen jet.
Figure 3-31 Shape of the combustion flame for shrouded oxygen jet.

Figure 3-32 Shape of the combustion flame for shrouded nitrogen jet.

Figure 3-31 and 3-32 shows the shape of combustion flame for both the oxygen and nitrogen jet. The maximum flame temperature is different for the two cases. As expected, the flame maximum temperature is higher for the oxygen jet because of the availability of extra oxygen. In case of shrouded oxygen jet, Figure 3-31 shows two combustion flames just after the exit from the nozzle. This is because oxygen is supplied from both the central Laval nozzle and outer ring of holes as shown in Figure 2-10 and fuel is injected from the inner ring of holes. Combustion occurs from both
sides of the fuel stream and the two flames merge together to a single flame downstream of their initial reaction zone. The flame temperature becomes maximum at around $\frac{x}{d_e} = 10$. The axial velocity distribution in the previous section shows that, the coherent length of the shrouded oxygen jet is higher than the shrouded nitrogen jet. The reason is the secondary flame which is generated at the periphery of the shrouded oxygen jet along with the primary combustion flame. The combustion that occurs in the shear mixing layer acts as a suppressant that delays the mixing of the central oxygen jet with the surrounding ambient. But in the case of shrouded nitrogen jet this secondary flame structure cannot form. The flame temperature for shrouded nitrogen jet reaches maximum just after the exit from the nozzle as shown in Figure 3-32. Then due to the high suction effect of the supersonic jet, the flame moves towards the central supersonic jet and propagates along with it.

The predicted maximum combustion flame temperature fluctuated by 4% throughout the simulation. It varied with time from around 3450K to 3600K for supersonic oxygen jet which represents the actual turbulent combustion scenario [105, 106]. For the supersonic nitrogen jet it varied from 2400K to 2500K. The predicted flame temperature distributions that are shown in Figure 3-31 and 3-32 are instantaneous values.

### 3.5.6.4 Vorticity And Turbulent Shear Stress Distributions

The vorticity is a measure of the rotation of a fluid element as it moves in the flow field. Vorticity is also a measure of mixing among the fluids. The higher the vorticity, the greater is the mixing. In Cartesian coordinates, the vorticity vector is [11]:

$$\zeta = \nabla \times \mathbf{U}$$  \hspace{1cm} 3-34

Where $\zeta$ is the vorticity vector $(s^{-1})$ and $\mathbf{U}$ is the velocity vector. When the supersonic jet passes through the relatively still air, rotational flow is developed at the periphery of the jet due to large velocity gradient at that region. Figure 3-33 and 3-34 shows the
predicted vorticity contour of the supersonic oxygen jet with and without shrouding flame respectively. For conventional jet, the vorticity region merges to the jet centreline more quickly compared to the supersonic jet with shrouding flame. This occurs because the shrouding flame delays the mixing of the central supersonic jet with the surroundings. Figure 3-35 shows the vorticity magnitude in radial direction at $\frac{x}{d_e} = 1, 3, 8$ and 12. The figure shows that the shrouding flame has shifted the vorticity region far from the jet periphery (Radial distance/ $d_e = 0.5$). With the increasing distance from the nozzle exit plane, the vorticity region is gradually approaching to the jet centreline and the shrouding flame is delaying the merging of the vorticity region with the jet centreline. For example, at $\frac{x}{d_e} = 12$ the vorticity region extends to the jet centreline in case of conventional oxygen jet whereas the vorticity region is still at the jet periphery of the shrouded oxygen jet. In the case of shrouded jet, additional vorticity regions are created by the shrouding gases just after the exit from the nozzle as shown in Figure 3-35 ($\frac{x}{d_e} = 1$). But the magnitude of the vorticity becomes negligible with the increasing distance from the nozzle exit plane.

Figure 3-33 CFD plot of vorticity contour for conventional oxygen jet.
Figure 3-34 CFD plot of vorticity contour for shrouded oxygen jet.

Figure 3-35 Radial distribution of vorticity magnitude at different axial locations for both conventional and shrouded oxygen jet.
Figure 3-36 and 3-37 shows the turbulent shear stress distribution of the supersonic oxygen jet for both the shrouding and non-shrouding cases respectively. For the shrouded oxygen jet, the value of maximum shear stress in the shear layer is approximately half of that of conventional jet. The shrouding combustion flame reduces
the density of the gases surrounding the main supersonic jet which in turn reduces the viscosity and turbulent shear stress in the shear layer. The reduced turbulent shear stress then delays the mixing of supersonic oxygen jet with the surroundings which in turn increases the potential core length of the jet. With the shrouding flame, the shear mixing layer merges with the jet centreline at around 40 nozzle exit diameter which is approximately equal to the potential core length of the coherent supersonic oxygen jet. For a conventional jet, the shear mixing layer merges with the jet centreline at around 10 nozzle exit diameter showing the end of potential core region of the jet.

### 3.5.6.5 Species Mass Fractions

Figure 3-38 shows the mass fraction of the oxygen along the central axis of the supersonic oxygen jet for both the shrouding and non-shrouding case. After the potential core region, the mass fraction of oxygen in the central jet starts to decrease and becomes equal to the ambient oxygen mass fraction. In steelmaking, oxygen is used to

![Figure 3-38 Predicted axial mass fraction distributions at the jet centreline of conventional and shrouded oxygen jet.](image)

Figure 3-38 Predicted axial mass fraction distributions at the jet centreline of conventional and shrouded oxygen jet.
refine the liquid iron into steel and knowledge of oxygen mass fraction distribution at the liquid metal interface is important for calculating the iron oxidation and decarburization rate. Also the higher the oxygen content at the impact area, the greater is the temperature developed at the impact zone [107].

Figure 3-39 shows the radial profile of CO$_2$ mass fraction for supersonic shrouded O$_2$ jet at different axial locations: $\frac{x}{d_e} = 1, 3, 8$ and 12. This figure shows two peaks of CO$_2$ mass fraction at $\frac{x}{d_e} = 1$ and 3 and only one peak at $\frac{x}{d_e} = 8$ and 12. This is because of the formation of two combustion flames just after the exit of the nozzle as discussed earlier. When the two flames merge together, radial distribution of CO$_2$ mass fraction shows only one peak. Figure 3-40 shows the radial profile of the mass fraction of CO$_2$ at the same axial locations for supersonic coherent nitrogen jet. As expected there is only one peak of CO$_2$ mass fraction because combustion occurs only on one side of the fuel stream.

Figure 3-39 Predicted radial distributions of the CO$_2$ mass fractions at different axial locations for shrouded oxygen jet.
Figure 3-40 Predicted radial distributions of the CO$_2$ mass fractions at different axial locations for shrouded nitrogen jet.

Figure 3-41 shows the calculated CO$_2$ mass fraction distribution for supersonic coherent O$_2$ jet. The figure shows that the CO$_2$ mass fraction is higher in the vicinity of combustion flame because CO$_2$ is the product of combustion. The mass fraction of H$_2$O shows a similar trend to that of CO$_2$.

Figure 3-41 CFD plot of the CO$_2$ mass fractions for shrouded oxygen jet.
3.5.7 Conclusions

CFD simulation of the supersonic oxygen and nitrogen jet with and without shrouding flame has been carried out. The present study showed that the shrouding combustion flame reduces the entrainment of the surrounding gas to the central supersonic jet which results in low spreading rate for the coherent supersonic jet. It also reduces the magnitude of turbulent shear stress in the shear layer which in turn delays the mixing of the supersonic jet with the surroundings. As a result, the potential core length of the supersonic coherent jet is increased compared to the conventional jet. The potential core length of the shrouded oxygen jet is over 4 times higher than the conventional oxygen jet. At steelmaking temperatures, the potential core length of the coherent supersonic oxygen jet is 1.4 times higher than that at room ambient temperature. In case of the shrouded nitrogen jet, the potential core length is over 3 times higher than the conventional nitrogen jet. The CFD results showed good agreement with the experimental data.

The present study only considered the one step complete combustion between CH\textsubscript{4} and O\textsubscript{2}. In reality, this combustion occurs in a number of steps. Apart from CO\textsubscript{2} and H\textsubscript{2}O, some other minor species like CO, H\textsubscript{2}, OH is also produced [84]. In case of coherent supersonic oxygen jet, the CO produced by incomplete combustion of CH\textsubscript{4} gas reacts with the O\textsubscript{2} which also creates a flame surrounding the jet and affects the potential core length of the jet. Further work is required to incorporate the multi-step combustion reaction.

The CFD model was validated only against experimental velocity distribution data. No experimental data is available in the literature for the flame temperature, mass fraction of different species to compare the CFD results against the experimental data. Hence, more experimental study is required to establish a more rigorous CFD model of the coherent supersonic jet.

The present study can provide some useful insights into coherent jet technology. The shape and temperature of the combustion flame is important for coherent jets and this study shows that the combustion flame temperature varies significantly when different gases are used as central supersonic jet. The present model can be used to predict the location of the hot spots of the combustion flame and the impact velocity distribution.
for different blowing conditions. The impact velocity of the gas jet on liquid surface will be higher if the potential core length is increased, which should also increase the droplet generation rate [7] though it is claimed that coherent jets produce less splashing [19, 24]. This model can also provide the distribution of mass fractions of different species which is very important for the process. The mass fraction of different species inside the furnace affects the partial pressure of the gases inside the furnace which in turn influences the kinetics of the reactions inside the furnace. The model developed in the present study should be helpful in determining the optimum flow rate of the shrouding gas and in designing more efficient coherent jet nozzles.
Chapter 4

4. Modelling Of Top Jet-Liquid Interaction
4.1 Introduction

In the present chapter, the modelling of the supersonic jet impingement on a liquid surface has been described. From the literature review, it was found that all the jet-liquid interaction modelling have been done either at low velocity of the jet (less than 100 m/s) or assuming the supersonic jet incompressible. In the present study, a new approach for modelling the shrouded supersonic jet impingement on a liquid free surface has been presented where two different computational domains were used, 1) single phase domain and 2) multiphase domain. In the single phase domain, the compressible gas phase was solved and the gas–liquid interaction has been simulated in the multiphase domain. The effect of shrouding gas flow rates on the depth of penetration and liquid phase velocity has been investigated. Then, the mechanism of droplets generation has been discussed in detail using the CFD results. The limitation of blowing number theory in calculating the droplet generation rates during the penetrating cavity mode has been investigated.

4.2 Computational Model

The computational model for solving the high speed, compressible gas jet have been described in section 3.2 and hence will not be repeated here. Therefore, in this section, the numerical modelling technique of multiphase flow has been presented only.

4.2.1 Assumptions

The following assumptions have been made while developing this computational model:

- Liquid phase is incompressible and Newtonian. The gas phase is compressible in the gas only computational domain.
- The flow is symmetric with respect to the vertical plane.
- The gas phase is incompressible in the multiphase domain. The available relevant studies[70, 108] on supersonic jet impingement considered the gas
phase incompressible even at 2.1 Mach number. The present assumption is much more realistic where gas is assumed incompressible only in the multiphase domain when the Mach number becomes less than 1. The errors incurred due to this assumption are evaluated and discussed in the results and discussion section.

4.2.2 Free Surface Modelling

In Eulerian grid, it is necessary to compute the flow of fluid through the mesh. This convective flux calculation requires an averaging of the flow properties. Convective averaging results in smoothing of all variations in flow quantities, and in particular, a smearing of the surfaces of discontinuity such as free surfaces. The only way to overcome this loss in resolution for free boundaries is to introduce some special treatment that recognizes a discontinuity(interface between different fluids) and avoids averaging across it [68].

A short review of different interface tracking method is given below:

4.2.2.1 Height Function

In this method, the vertical height of the free boundary from a reference line is defined as a function of the position along the reference line. For example, in a rectangular mesh of cells of width $\partial x$ and height $\partial y$, the vertical height $h$ of the free boundary can be defined above the bottom of the mesh in each column of cells. This would approximate a curve $h = f(x, t)$. This method does not work well when the boundary slope $\frac{dh}{dx}$ exceeds the mesh cell aspect ratio $\frac{\partial y}{\partial x}$ and does not work at all for multiple valued surfaces having more than one y value for a given x value. So many simple shapes such as bubbles, droplets cannot be treated. However, when it can be used, this method is extremely efficient requiring only a one dimensional storage array to record the surface height value [68]. In case of a free fluid boundary, the time evolution of the height function is governed by following kinematic equation,
Where \((U,V)\) are fluid velocity components in the \((x,y)\) coordinate directions \([68]\).

### 4.2.2.2 Marker and Cell Method

This method was introduced by Harlow and Welch \([109]\) where marker particles are spread over all fluid occupied regions. Each particle moves with the fluid velocity at its location. A mesh cell containing markers, but having a neighbouring cell with no markers, is defined as containing a free surface. The actual location of the free surface is determined by some additional computation based on the distribution of markers within the cell. This eliminates some problems associated with intersecting surfaces but the particle movement requires extra computational time and storage.

### 4.2.2.3 Level-Set Method

The Level-Set Method was first proposed by Osher and Sethian \([110]\). In this method a Level-Set function \(\Phi\) is defined whose value is zero at the actual interface position. This Level-Set function is assumed to take positive values inside the region under the interface and negative values outside. The advantage of Level-Set Method is that one can easily follow the shapes that changes topology. There is no mathematical difficulty in calculating local curvature and surface normal. The physical properties are usually averaged by a smoothed Heaviside function across the boundary:

\[
\rho(\Phi) = \begin{cases} 
\rho_1 + (\rho_2 - \rho_1)H & \text{For inner phase} \\
\frac{1}{2}(\rho_1 + \rho_2) & \text{For the boundary}
\end{cases}
\]

Momentum is solved with the conventional method and the Level-Set functions are advected with the following equation:
\[ \frac{\partial \phi}{\partial t} + U \cdot \nabla \phi = 0 \]

Where \( U \) is the velocity vector \((m/s)\).

4.2.2.4 Volume of Fluid Method

The volume of fluid method (VOF) was first introduced by Hirt and Nicholes [68]. In this approach, the motion of the interface is tracked by the solution of a scalar transport equation for a phase indicator field that is discontinuous at the interface and uniform elsewhere. This method can predict sharp interfaces between two or more immiscible phases and is widely used for its accuracy. Therefore, the VOF method has been adopted in the present study for the modelling of jet-liquid interaction.

4.2.3 Governing Equations of VOF Model

In case of incompressible flows, the governing equation for the volume fraction of tracked phase can be written in conservative form as:

\[ \frac{\partial \alpha}{\partial t} + \nabla \cdot \alpha \mathbf{U} = 0 \]

In the present study, \( \alpha \) is the volume fraction of liquid phase. Since, there are only two fluids in the domain and sum of all volume fractions in every cell must be one, the volume fraction of the gas becomes \((1 - \alpha)\). If the control volume is full of liquid, then \( \alpha = 1 \). Similarly, \( \alpha = 0 \) indicates the presence of gas. A value of \( \alpha \) between 0 and 1 defines the mixture of gas and liquid.

In VOF method, the different fluids are modelled as single continuum obeying the same set of governing equations, with the different fluids identified locally by a volume fraction field as discussed above. The governing equations for mass and momentum conservations are:
• Mass conservation equation

\[ \nabla \cdot \mathbf{U} = 0 \quad 4-4 \]

• Momentum conservation equation

\[ \frac{\partial \rho_m \mathbf{U}}{\partial t} + \nabla \cdot (\rho_m \mathbf{U} \mathbf{U}) = -\nabla P + \nabla \cdot (\tau + T^t) + \rho_m \mathbf{g} + f_\sigma \quad 4-5 \]

Where \( P \) is the pressure(\( Pa \)), \( \mathbf{g} \) is gravity vector (\( ms^{-2} \)) and \( f_\sigma \) is the force per unit volume due to surface tension.

The shear stress, \( \tau \), equals:

\[ \tau = \mu_m \left[ (\nabla \mathbf{U} + \nabla \mathbf{U}^T) - \frac{2}{3} \nabla \cdot \mathbf{U} \right] \quad 4-6 \]

Where \( \mu_m \) is the molecular viscosity(\( N - s/m^2 \)). Reynolds stress, \( T^t \), equals:

\[ T^t = \mu_t \left[ (\nabla \mathbf{U} + \nabla \mathbf{U}^T) - \frac{2}{3} \nabla \cdot \mathbf{U} \delta_{ij} \right] - \frac{2}{3} \rho_m k \delta_{ij} \quad 4-7 \]

Where \( \mu_t \) is the turbulent viscosity(\( N - s/m^2 \)), \( k \) is the turbulent kinetic energy(\( m^2 s^{-2} \)) and \( \delta_{ij} \) is the Kronecker delta (\( \delta_{ij} = 1 \), if \( i = j \) and \( \delta_{ij} = 0 \), if \( i \neq j \)). Modelling of turbulence parameters have been described in next section.

The density \( \rho_m \) and viscosity \( \mu_m \) of the fluid appearing in the transport equations are determined by weighted average in each control volume.
The surface tension force was calculated using the Continuum Surface Force (CSF) model developed by Brackbill et al.[74] where the surface tension effect is treated as body force:

\[ f_\sigma = \int_V \sigma \kappa \nabla \alpha dV = \sigma \kappa_p (\nabla \alpha_p) V_p \]  

Where \( V_p \) is the volume of cell \( p \) and \( \kappa_p \) is the curvature calculated by:

\[ \kappa_p = -(\nabla \cdot \hat{n}) = -\left[ \frac{v_a}{|v_a|} \right]_p \]  

Where \( \hat{n} \) is the unit normal to the surface.

The effect of wall adhesion was taken into account for the fluids which were in contact with the wall boundaries. This is done by adjusting the unit normal vector, \( \hat{n} \), to the interface according to the contact angle \( \theta_w \) (The angle between the wall and the tangent to the interface at the wall, measured inside the tracked phase). At any point on the wall, the unit normal vector can be expressed as:

\[ \hat{n} = \hat{n}_{wall} \cos \theta_w + \hat{n}_t \sin \theta_w \]
Where \( \hat{n}_{wall} \) is the unit wall normal directed into the wall and is computed from the geometry, whereas \( \hat{n}_e \) lies in the wall and is normal to the contact line between the interface and the wall. The value of \( \hat{n}_e \) is computed from the known volume fraction field by applying the symmetry condition at the wall boundary in question. Once \( \hat{n} \) is evaluated from equation (4.12), it is substituted in the equation (4.11) as a boundary condition for the cells near the wall. This change in curvature, in turn, modifies the surface tension near the wall [111].

As the VOF method solves one set of momentum, mass and other scalar equations, it is efficient in terms of time and memory required.

### 4.2.3.1 Advection of VOF Function \( \alpha \)

The discretization of equation (4.3), which governs the motion of the interface, is the key to the accurate capturing of the interface. The advection of volume fraction needs proper modelling to move the interface without diffusing, dispersing or wrinkling it. Many methods of VOF advection are available in the existing literatures which are described below.

Hirt and Nichols [68] proposed a donor-acceptor method for flux approximation. This method uses information about \( \alpha \) downstream as well as upstream of a flux boundary to compute the amount to be fluxed. For fluxes in a direction parallel to the surface, upstream fluxes are used. For fluxes in a direction perpendicular to the surface, downstream fluxes are used. After the advection calculation, a pass is made through the mesh to reset the values of \( \alpha \) less than zero back to zero and values of \( \alpha \) greater than one back to one. The position of interface within a cell is specified as either horizontal or vertical depending on the slope of the surface. The surface slope is determined by using volume fraction information in the neighborhood cell. After determining the surface slope a line is constructed in the cell with the correct amount of fluid volume lying on the fluid side.

In Hirt-Nichols donor-acceptor method, first order upwind flux produces diffusion error and downwind flux produces instability [112]. To eliminate the diffusiveness and instability, Rudman [112] proposed a flux corrected transport (FCT-VOF) in VOF
method. In this method, an intermediate value of volume fraction is defined first using the advection scheme. Then an anti-diffusive flux is defined to correct the numerical diffusion. A correction factor is used to limit the anti-diffusive flux. Finally, using the corrected anti-diffusive flux the values of volume fraction is obtained. The FCT scheme is non-diffusive, but it creates areas of unphysical jetsam and flotsam.

In Youngs’ VOF method [72], portion of interface in each cell is approximated by a straight line. The interface slope in a cell is determined by using volume fraction information in eight surrounding cell and the cell under consideration. The position of the interface is then adjusted so that it divides the cell into two areas which match the two volume fractions. This method is also known as Piece-wise Linear Interface Construction (PLIC) method. Advection of fluid is done separately in X and Y directions.

Rudman [112] compared the different VOF advection method for different flow conditions. Youngs’ method was found to give the most accurate results in all flow condition.

In the present study, the CICSAM (Compressive Interface Capturing Scheme for Arbitrary Meshes) developed by Ubbink and Issa [113] was used. It makes use of the NVD (Normalized Variable Diagram) and switches between the CBC (Convection Boundedness Criteria) and UQ (Ultimate-Quickest) schemes to maintain the boundedness and sharpness of the interface. Although this scheme is diffusive compared with Youngs’ method [72], the CICSAM scheme avoids the need of operator splitting and interface reconstruction. This method is robust and can be employed to unstructured meshes with arbitrary shape [111].

### 4.2.4 Turbulent Flow Modelling

The two equation $k - \varepsilon$ turbulence model with compressibility correction [17], described in section 3.3.4, was used to model high speed gas jet in the single phase domain at room ambient temperature. To describe the turbulence resulting from the jet liquid interaction in case of multiphase domain, the standard $k - \varepsilon$ turbulence model was used [14].
Turbulence kinetic energy equation for VOF model:

\[
\frac{\partial (\rho_m k)}{\partial t} + \nabla \cdot (\rho_m \bar{u} \bar{u} : \nabla U_k) = -\rho_m \bar{u} \cdot \nabla U_k + \frac{\mu_t}{\rho_m \sigma_\rho} \nabla \rho_m g + \nabla \cdot \left[ (\mu_m + \frac{\mu_t}{\sigma_k}) \nabla k \right] - \rho_m \varepsilon + K_{kl}
\]  

4-13

Turbulence dissipation equation for VOF model:

\[
\frac{\partial (\rho_m \varepsilon)}{\partial t} + \nabla \cdot (\rho_m \bar{u} \bar{u} : \nabla U_k) = -\rho_m \varepsilon C_{\varepsilon 1} \frac{\mu_t}{\rho_m \sigma_\rho} \nabla U_k - C_{\varepsilon 3} \frac{\mu_t}{\rho_m \sigma_\rho} \nabla \rho_m g + C_{\varepsilon 4} \rho_m \varepsilon \nabla U - \nabla \left( \left( \mu_m + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right) - C_{\varepsilon 2} \rho_m \frac{\varepsilon^2}{k} + D_{kl}
\]  

4-14

Where \( K_{kl} \) and \( K_{kl} \) is the interfacial exchange of kinetic energy and dissipation rate. The turbulent viscosity, \( \mu_t \), is modeled as:

\[
\mu_t = C_\mu \rho_m \frac{k^2}{\varepsilon} \]  

4-15

The closure coefficients of equation (4.13), (4.14) and (4.15) are similar to those described in section 3.3.4. In the present study, turbulence was assumed homogeneous in the two phases.

### 4.2.5 Computational Domain

The computational domain with boundary conditions is shown in Figure 4-1. Two
different computational domains were used for the simulation of shrouded supersonic jet impingement on liquid surface. In the single phase domain, the flow of supersonic jet was calculated, and in the multiphase domain the gas-liquid interaction was simulated. The exit plane of the nozzle was assumed to be the supersonic inlet as shown in Figure 4-1. The exit diameter of the nozzle was 9.54mm. The annular area for the shrouding gas flow was 2mm wide and located at a radius of 8mm from the nozzle centreline. The size of the single phase domain was 100 nozzle exit diameters downstream from the inlet plane and 15 nozzle exit diameters normal to the jet centreline. The total height and
radius of the multiphase domain were 1450mm and 575mm respectively which were selected according to the experimental study. The height of the liquid surface in multiphase domain was 900mm from the bottom as shown in Figure 4-1. The distance between the supersonic jet inlet plane and the water surface (known as lance height) was 250mm. The computational domains were two dimensional axisymmetric wedge shaped with only one cell in circumferential direction. The grid density was high along the jet centreline because of the high speed jet and at the jet-liquid interface region for accurately capturing the deformation of the interface.

4.2.6 Boundary Conditions

4.2.6.1 Single Phase Domain

The gas jet was assumed to have a uniform velocity magnitude at the nozzle exit. The stagnation pressure boundary condition was imposed at the inlet. The values of stagnation pressure and Mach number were defined. The turbulent kinetic energy at the nozzle inlet was calculated from the following equation [92]:

\[ k = 1.5(U_0 I)^2 \]  

Where \( I = \frac{u}{\bar{u}_0} \) is the turbulence intensity at the nozzle outlet and is assumed as 10% of the nozzle exit velocity. For shrouding gas inlet, the mass flow rate boundary condition was used. At the outlet, the static pressure boundary condition was used. The value of the static pressure was set equal to the atmospheric pressure. The no slip boundary condition for the velocities was used at the solid impermeable smooth wall. In order to model the near-wall effects (e.g. viscous damping) close to the wall, the compound wall treatment (CWT) (known as hybrid wall function in AVL Fire) approach [114] was used. It is a blending of integration up to the wall approach (ITW) and generalized wall function approach (GWF). The CWT approach can be described by following equation:
Where $E = 9.8$ is a constant, $\kappa = 0.41$ is a Von-Karman’s constant, $P_r$ is the Prandtl number, $U^+$ and $y^+$ are the dimensionless velocity and distance from the wall respectively. The subscript “$p$” denotes the first grid point near the wall and $\mu_w$ is the wall viscosity. The CWT approach provides the generalized wall function for large values of $y^+$ as well as integration up to the wall for the very small values of $y^+$. The advantage of the CWT approach is that this approach can provide adequate boundary conditions in complex non-equilibrium flows irrespective of whether the wall-nearest grid node lies in the viscous sub-layer, fully turbulent wall zone, or in the buffer region in between whereas in wall function approach the first grid point should always lie in the fully turbulent region [114].

### 4.2.6.2 Multiphase Domain Boundary Condition

In the multiphase domain, the velocity boundary condition was used at the inlet. From the single phase domain, the magnitude of velocity, turbulence kinetic energy and turbulence dissipation rate at a distance of 0.2m (dotted line in the Figure 4-1) was stored in a separate file and then used in the inlet boundary condition of multiphase domain through a user defined subroutine `usebnd.f`. All other boundary conditions at outlet, symmetry plane and wall were similar to those described in case of single phase domain. The values of the boundary conditions are listed in Table 4-1:
Table 4-1 Boundary conditions for modelling of top impinging jet.

<table>
<thead>
<tr>
<th>Name of the Boundary Conditions</th>
<th>Type of Boundary Conditions</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Supersonic inlet</td>
<td>Stagnation pressure</td>
<td>896320 Pa</td>
</tr>
<tr>
<td></td>
<td>Mach number</td>
<td>2.1</td>
</tr>
<tr>
<td></td>
<td>Total temperature</td>
<td>298K</td>
</tr>
<tr>
<td>Shrouding gas inlet</td>
<td>Mass flow rate</td>
<td>Lower $3.836 \times 10^{-6}$ kg/s,</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Higher $7.1 \times 10^{-6}$ kg/s</td>
</tr>
<tr>
<td></td>
<td>Total temperature</td>
<td>298K</td>
</tr>
<tr>
<td>Outlet</td>
<td>Static pressure</td>
<td>100000 kPa</td>
</tr>
<tr>
<td>Symmetry</td>
<td>Symmetry</td>
<td>-</td>
</tr>
<tr>
<td>Wall</td>
<td>No slip</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Smooth wall</td>
<td>-</td>
</tr>
</tbody>
</table>

The physical properties of the two fluids, used in the present study at 298K, are presented in Table 4-2:

Table 4-2 Physical properties of the fluids.

<table>
<thead>
<tr>
<th>Properties</th>
<th>Fluids</th>
<th>Gas</th>
<th>Liquid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m$^3$)</td>
<td></td>
<td>1.2</td>
<td>997</td>
</tr>
<tr>
<td>Viscosity(Ns/m$^2$)</td>
<td></td>
<td>$1.824 \times 10^{-5}$</td>
<td>0.000798</td>
</tr>
<tr>
<td>Surface tension (N/m)</td>
<td></td>
<td>-</td>
<td>0.071</td>
</tr>
</tbody>
</table>

4.2.7 Computational Procedure

The computational model was developed using the finite volume based commercial CFD software AVL Fire Version 2009.2. The computations were carried out in an Intel Xeon work station with 2.67GHz quad core processor and 8GB of Ram. For the single phase domain, the unsteady, compressible continuity, momentum and energy equations were solved using segregated solver to calculate the pressure, velocity, temperature and density. Pressure-velocity coupling was achieved using the SIMPLE algorithm [94]. Because of the high speed jet, very small time step of $1 \times 10^{-4}$s was used to obtain a stable solution. Then the values of velocity, turbulence kinetic energy and dissipation rate at a predetermined location (shown by the dotted line in Figure 4-1) were stored in a dat file. This file was then called in the multiphase domain using a user defined
subroutine at the inlet face. In the multiphase domain, both the gas and liquid phases were assumed incompressible. The volume fraction equation (4.3) was solved first at the beginning of each time step. The volume fraction field was then used to compute the mixture viscosity and density using equations (4.8) and (4.9). Then the incompressible continuity (4.4) and momentum (4.5) equations were solved using SIMPLE algorithm which executes a sequence of predictor-corrector steps utilizing a pressure equation that is derived from a combination of equation (4.4) and (4.5). For the discretization of momentum and pressure correction equation, the AVL SMART scheme and central differencing scheme (CDS) was used. The value of the time step, used in the simulation, was $1 \times 10^{-4}$s up to 0.1s and then 0.0001s was used for the rest of the simulation. The simulations were run for 10s to guarantee the stability in the solution. The total number of cells in the single phase and multiphase domains were 20345 and 68145 respectively. The under-relaxation factors used in the simulation are presented in Table 4-3. A grid independency test was carried out to analyse the sensitivity of the solution to the grid size which is presented in section 4.3.3.

Table 4-3 Under-relaxation factors for gas impingement study

<table>
<thead>
<tr>
<th>Equation</th>
<th>Under-relaxation factors</th>
</tr>
</thead>
<tbody>
<tr>
<td>Momentum</td>
<td>0.3</td>
</tr>
<tr>
<td>Pressure correction</td>
<td>0.1</td>
</tr>
<tr>
<td>Turbulence Kinetic energy</td>
<td>0.4</td>
</tr>
<tr>
<td>Turbulence Dissipation rate</td>
<td>0.4</td>
</tr>
<tr>
<td>Volume fraction</td>
<td>1</td>
</tr>
</tbody>
</table>

4.3 Results and Discussions

4.3.1 Shrouding Gas Jet

Figure 4-2 shows the effect of shrouding gas flow rate on the axial velocity distribution of the supersonic jet. At higher shrouding gas flow rate, the coherent region of the supersonic jet was found to increases which was expected because at higher shrouding gas flow rate, the spreading rate of the turbulent jet decreases [13].
Figure 4-2 Axial velocity distributions of supersonic jet at two different shrouding gas flow rates.

As there was no experimental data available to validate the axial velocity distribution for two different shrouding gas flow rates, these are compared with the well-known semi-analytical equations:

\[ \frac{u}{u_0} = K_2 \frac{d_e}{X} \]

Meidani et al. [29] showed that the values of \( K_2 \) depends on the shrouding gas flow rates. They reported two values of \( K_2 = 10.7 \) and 11.5 from their experimental study for the lower and higher shrouding flow rates respectively. Using the appropriate value of \( K_2 \), the axial velocity distributions were calculated for the two different shrouding gas flow rates using equation (4.20) and were also plotted in Figure 4-2. The Figure shows that the CFD results are in good agreement with the calculated velocity distribution.
from equation (4.20). The average percentage of deviation is less than 9% and 8% for the higher and lower shrouding flow rates.

Figure 4-3 Axial dynamic pressure distribution of supersonic jets at two different shrouding gas flow rates.

Figure 4-3 shows the dynamic pressure distribution of the supersonic jet at two different shrouding gas flow rates. In the experimental study [29], the gauge stagnation pressure of the supersonic jet for both the higher and lower shrouding case was measured at a distance of 152mm from the nozzle exit. The gauge stagnation pressure is actually the dynamic pressure of the jet. Because, the stagnation pressure is the summation of static and dynamic pressure, and in the experimental study, the static pressure was equal to the atmospheric pressure. The measured pressure data falls exactly on the computed axial dynamic pressure curve for both the higher and lower shrouding flow rates as shown in Figure 4-3.
4.3.2 Gas Jet Penetration

The deformation on liquid surface, created by the impinging jet, was calculated for the two different shrouding gas flow rates through accounting the force balance at the interface. The depth of penetration created by the impinging gas jet was higher in case of higher shrouding gas flow rate compared with the lower shrouding gas flow rate. Table 4-4 shows both the predicted and measured average penetration depths at different shrouding gas flow rates:

Table 4-4 Average penetration depth for different shrouding flow rates.

<table>
<thead>
<tr>
<th>Nozzle-bath distance</th>
<th>Shrouding flow rate 3.836×10⁶ kg/s</th>
<th>Shrouding flow rate 7.1×10⁶ kg/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>250 mm</td>
<td>CFD</td>
<td>Experiment</td>
</tr>
<tr>
<td>374 mm</td>
<td>546 mm</td>
<td>31</td>
</tr>
</tbody>
</table>

As expected, a deeper cavity was formed when shrouding gas flow rate was higher compared with the lower shrouding gas flow rate. Because, the higher shrouding flow results in an increase in the coherent region of the jet which in turn results in higher impact pressure on the liquid surface. Table 4-4 shows that the CFD model underpredicts the penetration depth by 31% and 32% in case of higher and lower shrouding flow rates respectively. One of the sources of error in the CFD model may be the assumption of incompressible gas jet in the multiphase domain. As discussed earlier, in the present study the compressible gas jet was solved first in the single phase domain. Then the properties (velocity, turbulent kinetic energy and dissipation rate) of the jet at a plane of 200mm from the nozzle exit were used as inlet boundary condition for the multiphase domain. After 200mm from the nozzle exit plane, the gas jet becomes subsonic but not incompressible. The Mach number of the jet becomes 0.9 and 0.8 for higher and lower shrouding flow rate respectively. But in case of multiphase flow, the gas jet was assumed incompressible which introduced error in the dynamic pressure calculation of the jet. For a compressible jet, the dynamic pressure is calculated by [115].
\[ P_d = \frac{1}{2} C_p \rho g U^2 \]

Where \( C_p \) is the pressure coefficient. The value of pressure coefficient can be determined from the following equation [115]:

\[ C_p = \frac{2}{\psi M a^2} \left( 1 + \frac{\psi^{-1}}{\psi} M a^2 \right)^{\psi^{-1}} - 1 \]

Where \( \psi = 1.4 \) is a constant [115]. Using the values of \( M a = 0.8 \) and 0.9, the pressure coefficient \( C_p \) becomes 1.17 and 1.22 respectively. But under the assumption of incompressible flow \( C_p = 1 \) was used in the present study. Hence, this assumption results in 17% and 22% error in dynamic pressure calculation in case of lower and higher shrouding flow rates respectively. The standard \( k - \varepsilon \) turbulence model, used in the multiphase flow simulation, might be another source of error in the penetration depth calculation. As discussed in section 2.3.2, the standard \( k - \varepsilon \) turbulence model underpredicts the depth of penetration [76]. An alteration of the constant \( C_{e2} \) of equation (4.14) may result in higher penetration depth of the jet [76]. But it was also reported [76] that the alteration in \( C_{e2} \) value changes the liquid flow pattern. Also, the CSF model which was used in the present study to model the momentum interfacial exchange is based on the simplified boundary conditions. Hwang and Irons [75] showed that the full stress boundary conditions at the gas liquid interface can predict the surface waves and cavity dimensions more accurately.

The droplets that are generated from the edge of the cavity sometimes re-circulate back into the gas jet and impinge on the liquid surface, particularly, in case of higher nozzle flow rate where the cavity operates in penetrating mode. These droplets provide extra momentum on the liquid surface and hence may result in higher penetration depth. As two different domains were used in the present study, the recirculation of droplets in the gas jets could not be accounted for fully which might be another reason for the underprediction of the penetration depths. Last, but not the least, is the numerical error.
involved in the simulation which may arise from the discretization of the partial
differential equation and also from the differencing scheme used in the simulation. A
fine grid was used to reduce discretization error in the simulation.

Although, the present approach underpredicted the depth of penetration, it showed an
improvement over the previous approach where supersonic jet was assumed
incompressible and a single domain was used to model the multiphase flow [70]. In the
present study, a simulation of jet impingement was carried out in a single domain
assuming the supersonic jet is incompressible. The average depth of depression was
found to be 190 mm having a 65% error when compared with experimental study.
Hence, the present study proves that the use of two different domains for supersonic jet
impingement study improves the prediction of cavity depths.

4.3.3 Grid Sensitivity Test

In order to study the grid sensitivity of the solution, calculations of depth of penetration

![Figure 4-4 Calculated depth of penetration for different grid levels.](image)

in case of lower shrouding flow rate was carried out using three different grid levels:
coarse grid (40410 cells), medium grid (68145 cells) and fine grid (102750 cells). The
calculated depths of penetration for all grid levels are shown in Figure 4-4. The
variation in the depth of penetration, calculated with coarse and medium grid, is within
10% and it becomes less than 1% between medium and fine grid. Hence, it can be said that the solution is not sensitive to the grid. The solution with fine grid takes longer time than the solution with medium grid. Therefore, considering the computational time, the results obtained with medium grid were used for analysis and discussion in this study.

### 4.3.4 Droplets Generation Study

It was reported in the literature [54] that the surface waves are generated within the crater during jetting and these waves then propagate across the whole surface of the liquid. At higher gas flow rates, these surface waves form finger like structure near the edge of the cavity, and finally, are torn off due to the unstable structure of these metal fingers [38]. This mechanism of droplets generation was observed in the numerical simulation carried out in the present study. Figure 4-5(a)-(d) shows the mechanism of droplets generation from the edge of the cavity during top jetting. Figure 4-5(a) shows the generation of a surface wave inside the cavity which is the first stage of droplets
formation. This surface wave then generates a metal finger near the edge of the cavity as shown in Figure 4-5(b). The metal finger has no structural stability and once it is accelerated upward, the high speed gas breaks the fingers off from the main liquid bath as shown in Figure 4-5(c). This is the end of one cycle of droplets generation mechanism. Figure 4-5(d) shows the formation of another surface wave inside the cavity which is the beginning of next cycle. In the present study, although the splashing of liquid was observed, it could not be quantified. Because, very fine mesh is required to simulate the individual droplets which will be computationally very intensive. But the predicted transient nature of the cavity agreed reasonably well with the experimental description provided by the previous researchers [38].
Figure 4-5 Mechanism of droplets generation.
In the present CFD study, the cavity shape was calculated from the force balance at the interface. The surface waves were generated and the liquid droplets were torn from the edge of the cavity due to the shearing effect. The molecular and turbulent shearing effect due to the difference in velocity magnitude between gas and liquid phase were taken into account through equation (4.5) to (4.7). Dogan et al.[99] reported that the liquid surface tension influences the droplet generation rate. In the present study, the effect of liquid surface tension on the jet liquid interaction was taken into account through equation (4.5) and (4.10).

4.3.5 Limitation of Blowing Number \((N_B)\) Theory

According to the Kelvin-Helmholtz theory, onset of splashing occurs when the gas-liquid interface becomes unstable. Subagyo et al.[7] reported that the gas-liquid interface becomes unstable when the blowing number \(N_B\) becomes greater than 1. They have proposed an equation for calculating Blowing number which was mentioned in equation (2.32) and is also presented here,

\[
N_B = \frac{\rho_g U_g^2}{2 \sigma_g \rho_l}
\]  

(4-23)

Where \(U_g\) denotes the critical gas jet velocity \((m/s)\) for the onset of splashing, related to the free turbulent jet velocity, \(U_j\), by the following equation:

\[
U_g = \eta U_j
\]  

(4-24)

Where \(\eta\) is a constant and was found to be in the range of 0.42 to 0.46 [7, 50] for top jetting case. In the present CFD model, the turbulent jet velocity \((U_j)\) and the critical gas velocity \((U_g)\) were defined at the jet liquid interface as shown in Figure 4-6. The
Figure 4-6 CFD prediction of the velocity distribution resulting from top jetting.

Figure 4-7 Variation of $\eta$ value with lance heights.
magnitudes of $U_j$ and $U_g$ were predicted at five different lance heights (250mm, 400mm, 500mm, 700mm and 800mm) and the value of $\eta$ was calculated for each cases. It was found from the present CFD model that $\eta$ value is a function of lance height and it increases with increasing lance height as shown in Figure 4-7. The figure shows that at 800mm lance height, the value of $\eta$ was in the range of the experimentally reported values. The $\eta$ values, reported in the literature, were determined at the onset of splashing where the cavity just transforms from the dimpling to splashing mode [50]. Figure 4-7 shows that the values of $\eta$ decreased with decreasing lance heights and became one-third of the reported value at 250mm lance height where the cavity was in penetrating mode. Hence, a constant $\eta$ value cannot be used at all lance height for calculating the droplet generation rates. It will over-predict the Blowing number and hence, the droplets generation rate if the cavity is in penetrating mode. This finding is in excellent agreement with the experimental study carried out by the present authors (chapter 5) where it was reported that if the lance height is lowered gradually, the Blowing number increases but the droplet generation rate decreases. The reason might be that the Blowing number was calculated using a constant value of $\eta$ in that experimental study and variation of $\eta$ value with lance heights was not taken into account.

Table 4-5 CFD prediction of the turbulent free jet ($U_j$) and critical gas velocity ($U_g$) at different lance heights.

<table>
<thead>
<tr>
<th>Lance height (mm)</th>
<th>$U_j$ (m/s)</th>
<th>$U_g$ (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>250</td>
<td>129.5</td>
<td>18.86</td>
</tr>
<tr>
<td>400</td>
<td>107.5</td>
<td>19.55</td>
</tr>
<tr>
<td>500</td>
<td>82.63</td>
<td>19.1</td>
</tr>
<tr>
<td>700</td>
<td>54.3</td>
<td>19.15</td>
</tr>
<tr>
<td>800</td>
<td>44.33</td>
<td>19.6</td>
</tr>
</tbody>
</table>

Table 4-5 shows the turbulent jet velocity ($U_j$) and the critical gas velocity ($U_g$) predicted from the present CFD model at different lance heights. It was very interesting
to notice that the values of $U_g$, which is responsible for shearing of the droplets from the liquid surface, remained almost constant at all lance heights for a particular jet inlet velocity. The extra turbulent jet velocity ($U_j$), in case of lower lance heights, was used for holding the cavity shape and stirring the bath. Then the question arises why the splashing rate increases with increasing lance heights from close to the liquid surface to a certain position and then again decreases with further increase in lance heights.

Figure 4-8 shows a typical variation of droplet generation rate with lance heights observed by the previous researchers [6, 9, 67]. Since the critical gas velocity ($U_g$) is independent of lance heights, the only contributing factor to the increment of droplet generation rate might be the change in cavity surface area. Table 4-6 shows the dimensions of the cavity at different lance heights. We assumed that if the ratio of cavity diameter to depth is less than one, it is operating in penetrating mode. Based on this assumption, it was found that at 250mm lance height the cavity was operating in penetrating mode and then it slowly transformed to splashing mode with the increase in
lance heights. The cavity depth decreases but the diameter increases which results in higher gas-liquid interfacial area and in turn increases the droplets generation rates. If the lance height is increased after a certain position (position 2 in Figure 3), the momentum exchange becomes the dominant factor over the cavity dimensions and as a result of that the droplet generation rate decreases because the jet momentum decreases with increasing lance heights. From the above discussions, it can be concluded that the blowing number theory can be used in the region of “2” to “3” of Figure 4-8 with proper \( \eta \) values but in the region of “1” to “2” the cavity surface area has to be taken into account.

Table 4-6 Cavity dimensions at different lance heights

<table>
<thead>
<tr>
<th>Lance heights (mm)</th>
<th>Cavity depth (mm)</th>
<th>Cavity diameter (mm)</th>
<th>Diameter to depth</th>
</tr>
</thead>
<tbody>
<tr>
<td>250</td>
<td>373</td>
<td>291</td>
<td>0.78</td>
</tr>
<tr>
<td>400</td>
<td>221</td>
<td>307</td>
<td>1.39</td>
</tr>
<tr>
<td>500</td>
<td>192</td>
<td>367</td>
<td>1.91</td>
</tr>
<tr>
<td>700</td>
<td>147</td>
<td>460</td>
<td>3.13</td>
</tr>
<tr>
<td>800</td>
<td>132</td>
<td>493</td>
<td>3.73</td>
</tr>
</tbody>
</table>

4.3.6 Fluid Flow in Liquid Bath

Figure 4-9 and 4-10 shows the volume fractions and velocity fields predicted by the model at time \( t=3.9 \) sec for both higher and lower shrouding flow case. In both cases, jet transfers momentum to the liquid bath and due to the shearing effect of the high speed gas jet, liquid free surface travels from the jet impingement point to the vertical wall where the flow turns down and returns to the impingement point creating a clockwise circulation loop “1” as shown in Figure 4-9 and 4-10. In case of higher shrouding flow rate case, liquid surface velocity on top of recirculation zone “1” was in the range of 0.2m/s compared with 0.15m/s for lower shrouding flow rate case. This indicates strong recirculation in the liquid field which in turn lowers mixing time in case of higher shrouding flow rate. This is in agreement with the experimental results of the Meidani et al.[29] where it was reported that mixing time is lower in case higher shrouding flow rate compared to the lower one. This occurs due to the higher momentum transfer associated with the supersonic jet with higher shrouding flow rate. In both cases, the liquid surface velocity was found to decrease as it travels from the jet impingement point to the vertical wall.
Figure 4-9(a) Volume fraction and (b) velocity vectors at time $t=3.9s$ for higher shrouding flow case.

Figure 4-10(a) Volume fraction and (b) velocity vectors at time $t=3.9s$ for lower shrouding flow case.
Zone “2” is the recirculation of deflected gas into the main jet. Sometimes liquid droplets, torn from the cavity edge, also re-circulate into the main jet and impinge on the liquid surface. The vortex in zone “3” creates the surface wave leading to the formation of metal fingers as discussed in section 4.3.4. Figure 4-9 shows some more vortexes in the gas-liquid emulsion zone. These recirculation zones occurred surrounding the liquid droplets in the emulsion phase. However, they were very random in nature and appeared randomly at different time steps.

### 4.3.7 Effect of Liquid Density

Increasing the density of liquid metal is predicted to reduce the depth of penetration in the liquid bath. In the present study, effect of liquid density on the depth of penetration was investigated by using different liquid densities ranging from liquid aluminium up to the molten steel. The depth of penetration results from the CFD model was then compared with the calculated depth of penetration using equation (2.20). In this equation, \( k_2 = 10.17 \) was used as discussed in section 4.3.1. The calculated depths of penetration at different liquid densities are presented in Figure 4-11. The figure shows

![Figure 4-11 Variation of depth of penetration with liquid density.](image)

---

154
that the depth of penetration predicted by CFD model is consistently lower compared with value calculated by equation (2.20) but follows the similar decreasing pattern. The reason behind the underprediction was discussed in section 4.3.2.

4.4 Conclusions

A computational fluid dynamics model has been developed to model a shrouded supersonic jet impingement on the liquid surface and validated against the relevant experimental data. Two different domains have been used to solve the problem of compressible jet impingement on incompressible liquid surface. The following conclusions have been reached from the current study:

- The coherent length of the supersonic shrouded jet increases with the increase of shrouding gas flow rates which results in higher depth of penetration on the liquid surface.
- The CFD model predicts the dynamic nature of the cavity and the mechanism of droplet generation which agrees well with the theoretical explanation.
- The \( \eta \) value in the Blowing number equation is a function of lance height and hence, a constant value of \( \eta \) cannot be used at all lance heights.
- The Blowing number theory fails to predict the droplet generation rate if the cavity is in penetrating mode.
- The cavity surface area might be the most influencing factor in the generation of liquid droplets. The higher the surface area, the more is the droplet generation rate. The cavity surface area increases when the cavity transforms from the penetrating to splashing mode with the increase in lance height. After a certain lance height, the momentum of the jet becomes the dominant factor.
- The impingement of supersonic jet creates circulation in the liquid bath. The velocity of the liquid free surface was higher in case of higher shrouding flow rates. This indicates stronger circulation as well as lower mixing time in case of higher shrouding flow rates.
Chapter 5

5. Experiment on Inclined Jetting And Splashing
5.1 Introduction

This chapter describes the physical modelling of inclined jetting and splashing inside steelmaking furnace. The effect of different operating conditions (lance angle, lance height and flow rates) on the depth of penetration and wall splashing rate is investigated. The variation of critical depth of penetration with lance inclination angle is discussed. A modification of the dimensionless Blowing number, which is a measure of the droplet generation rate, has been proposed including the effect of lance inclination angle.

5.2 Experimental Apparatus and Set-up.

The experimental study was carried out at Steel Research Centre in McMaster University, Canada. The experimental setup, used by McGee and Irons [53], was modified for the current study. A one-third scale thin slice water model was used to simulate the ArcelorMittal Dofasco’s EAF. Dynamic similarity between the one-third scale model and the real system was maintained through modified Froude number which was presented in equation (2-26). In the one-third thin slice model, the jet cannot expand freely and hence, penetrates more deeply compared with the one-third scale model. Consequently, the flow rates used in the slice model was reduced to provide the similar jet penetration to that for a three dimensional one-third scale model. The similarity calculations are presented in Table 5-1 [53].

Table 5-1 Dimensional analysis of the oxygen lance[53].

<table>
<thead>
<tr>
<th>Properties</th>
<th>Dofasco</th>
<th>One-third physical model</th>
<th>One-third thin slice model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bath/Gas</td>
<td>Liquid steel/oxygen</td>
<td>water/air</td>
<td>water/air</td>
</tr>
<tr>
<td>Liquid density (kg/m³)</td>
<td>7000</td>
<td>1000</td>
<td>1000</td>
</tr>
<tr>
<td>Gas density (kg/m³)</td>
<td>1.31</td>
<td>1.19</td>
<td>1.19</td>
</tr>
<tr>
<td>Lance diameter (mm)</td>
<td>42.5</td>
<td>14</td>
<td>14</td>
</tr>
<tr>
<td>Flow rate(Nm³/hour)</td>
<td>3500 to 5000</td>
<td>86-324</td>
<td>17 to 51</td>
</tr>
<tr>
<td>Velocity</td>
<td>685 to 979</td>
<td>156 to 223</td>
<td>31 to 92</td>
</tr>
<tr>
<td>Modified Froude number</td>
<td>211 to 430</td>
<td>211 to 430</td>
<td>8 to 73</td>
</tr>
</tbody>
</table>
Figure 5-1 The two dimensional thin slice model for the experimental study.

Figure 5-1 shows the thin slice model of the furnace which was 2080mm long, 1450mm deep and 25mm wide. The thin slice model was used to allow measurement of the penetration depth and observe the shape of the cavity more easily and accurately. Thin slice nature of the model introduces some artefacts; the jets penetrate more deeply than a 3D jet because they are constrained in the thin slice dimension; the jet model developed by McGee and Irons accounts for this phenomenon [53]. Three different 14mm inner diameter lance was used to inject air at three different lance angles on the water surface. In order to ensure fully developed turbulent flow inside the lance, the lance length was selected much longer than the required length. This is known as entry length [11]. According to Cengel and Cimbala [11] the minimum pipe length required to develop fully turbulent flow inside a pipe can be determined by following equation:

\[
\frac{L}{d} = 1.359Re^{0.25}
\]

(5-1)

Where \(L\) is the pipe length(\textit{m}) and \(d\) is the lance diameter(\textit{m}). The maximum velocity used in the present study was 76m/s. Therefore, the minimum pipe length required was
309mm. But the lances used in the present study were 655mm in length. After 600mm, the front side of each lance was bent in required angle for the experiment. The bending radius was kept high enough to minimize the bending loss. The other side of the lance was connected with a compressed air tank via transparent rubber tube. The commercial rotameter “KING” was used to measure the flow rate of air. The maximum capacity of the rotameter was 1.7Nm$^3$/min (60 scfm). The flow rates used in the experiment were ranged between 0.43 to 0.71Nm$^3$/min. A pressure gauge was used to keep the line pressure at 345kPa (50psi) throughout the experiment.

In the present study, water and air were used to simulate steel and oxygen respectively. A slag layer was not considered. When the gas jet impinged on liquid surface, it created a depression on the liquid surface and liquid droplets, generated from the cavity, splashed on the wall of the vessel. The depression depth and shape of the cavity was recorded by using Olympus C-7070 wide zoom camera with 7.1 megapixel resolution and 4x optical zoom. To illuminate the jet impingement area, a 200W lamp was used on the front lower side. The videos were recorded at 15fps (frames per second). Then by using “Photron Fastcam Viewer” software, the video recording was converted to 15 images per second. The images were then used to measure the cavity depth and diameter with the help of the grid attached on the side wall of the water model as shown in Figure 5-2. A schematic diagram of the experimental set up is shown in Figure 5-3.

Figure 5-2 Grid on the furnace wall to measure the depth of the cavity.
Figure 5-3 Schematic of experimental Set-up.

Figure 5-4 Splash collector used in the study.
5.3 Experimental Procedure

Splashing rates were measured by collecting the splashed liquid in eleven small plastic containers which were attached in an aluminium rod as shown in Figure 5-4. The splashed liquid was collected at a distance of 750mm (shown by dotted line in Figure 5-1) from the jet impingement point. The length, width and height of each plastic container were 40mm, 23mm and 25mm respectively. The plastic containers were numbered from 1 to 11 starting from the bottom (close to water surface). The gap between the bottom surface of first container and the undisturbed water surface was 100mm. The gaps between containers are shown in Figure 5-4. After the impingement of gas jet and commencement of splashing, the aluminium rod with the buckets was inserted in the thin slice model, at the position shown by dotted line in Figure 5-1 and taken out after certain time. The time of splash collection was varied from 10 seconds (High Flow rate) to 3 minutes (Low flow rate). The total volume of water in each bucket was then measured by pouring them into a measuring flask. Then the total volume of splashed liquid was divided by the time of splash collection to calculate the splashing rate in millilitres per second. Splashing rates were measured for different operating conditions which are summarized in Table 5-2. The flow rates were selected based on modified Froude Number similarity to model the Dofasco’s full scale furnace. The details of the similarity analysis were presented by McGee and Irons [53] in their paper. For each set of operating conditions (lance height, lance angle and flowrate), splashing rate was measured six times and the average was taken as the splashing rate for that particular operating condition. Only forward splashing rate was measured in the present study due to the thin slice nature of the model which is also the maximum splashing rate because in case of inclined jetting splashing rate is maximum in the forward direction directly in front of the lance. The error in the measurement of splashed liquid, penetration depth and time was ±0.2ml, ±5mm and ±0.05sec respectively. The method of calculating the error in the measurement of splashing rate has been described in Appendix A. The calculated maximum error in the measurement of splashing was 0.008ml/sec in case of 45 degree lance angle and 105mm lance height. Fine grid was used to measure the critical penetration depth. The accuracy of critical depth measurement was ±1mm. The present experimental study underpredicts the actual forward splashing rate because the droplets hitting the surface of the plastic bucket could not be collected in the plastic container for measurement.
Table 5-2 Operating conditions selected for the present study.

<table>
<thead>
<tr>
<th>Lance heights (m)</th>
<th>Angles (degree)</th>
<th>Flow rates (Nm$^3$/min)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.065</td>
<td>35</td>
<td>0.43</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.43</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.43</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
</tr>
<tr>
<td>0.105</td>
<td>35</td>
<td>0.43</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.43</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.43</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
</tr>
<tr>
<td>0.125</td>
<td>35</td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.145</td>
<td>35</td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.57</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
5.4 Results and Discussions

5.4.1 Effect of Lance Gas Flow Rates

Figure 5-5 shows that the higher the lance gas flow rate, the deeper the cavity which is expected because of the high dynamic pressure. McGee and Irons [53] showed that the depth of penetration is proportional to the square of flow rate which is not clear from Figure 5-5. More simulation data on depth of penetration is required to observe this relationship.

Figure 5-6 shows the wall splashing rate at different flow rates. As expected, splashing rates were higher at higher flow rates at all lance heights and angles. This is because at higher gas flow rates, the impact dynamic pressure as well as shear stress at the gas liquid interface is higher which in turn increases the droplet generation rate. But the slope of these curves was found to be different at different lance angles and lance heights. At 65mm lance height, the splashing rate increased dramatically with lance angle compared with 165mm lance height where the slope of the splashing curves did not vary significantly with lance angle. Also at 45 degree lance angle, splashing rate increased rapidly with flow rates compared to other lance angle at all lance heights. The reasons behind these observations are discussed in section 5.4.2.
Figure 5-5 Depth of penetration vs flow rate at 35 and 45 degree lance angles and 0.105m lance height. Data points are an average of 30 penetration depth measurements, the error bars are the standard deviation for the 30 measurements and were found to be approximately 10% of the average penetration depth.
Figure 5-6 Splashing rate vs flow rate at (a) 65 mm (b) 105 mm and (c) 165 mm lance heights. Data points are an average of 6 splashing rate measurements; the error bars are the standard deviation for the 6 measurements and were found to be approximately 9% of the average splashing rate.
5.4.2 Effect of Lance Angle

Figure 5-6 shows that at a constant flow rate and lance height, splashing rate increases with increasing lance angle from the vertical. This can be explained from the shape of the cavity formed during jetting. Figure 5-7 shows the shape of the cavity formed at two different operating conditions. At 35 degree lance angle, deeper cavity was formed compared to 45 degree lance angle and the droplets that were generated from the edge of the cavity, recirculates into the cavity which in turn reduced the forward splashing rate [38]. This is known as the penetrating mode [36] of operation. On the other hand, at 45 degree lance angle, the droplets that were generated from the edge of the cavity, moved forward and splashed on the wall of the furnace. This is known as the splashing mode [36] of operation. With increasing lance angle from vertical, cavity shape changed from the penetrating mode to splashing mode which in turn increased the rate of wall splashing. Figure 5-6 also shows that at constant flow rate, increasing rate of splashing with lance angle is different at different lance heights.
Figure 5-7 Cavity formed at 65mm lance height, 0.71Nm3/min flow rate and at a lance angle of (a) 35 degree and (b) 45 degree.

Figure 5-6 is modified and presented in Figure 5-8 to show the effect of lance angle on splashing rate. The figure shows that the rate of increase of forward splashing with lance angle is higher at 65mm and 105mm lance height compared with 165mm lance height. As discussed earlier, at 65mm lance height, the change in angle changed the impingement mode from penetrating to splashing which in turn increased the splashing rate. Also, the deflected gas velocity at the gas liquid interface after impingement was higher at the steeper lance angle which in turn increased the splashing rate because of high shear stress. This was also the case for 105mm lance height. But at 165mm lance height, the cavity was already in splashing mode at 35 degree lance angle. Droplets that were generating from the edge of the cavity were ejecting outside rather than recirculating inside the cavity. The change in lance angle from 35 degree to 45 degree increased the splashing rate only due to the increase in deflected gas velocity, as well as, the shear stress magnitude at the gas liquid interface, which has been described earlier. As a result, increasing rate of splashing with lance angle (slope of the curves) is higher at 65mm and 105mm lance height compared to 165mm lance height.
5.4.3 Effect of Lance Height

Figure 5-9 shows the change in splashing rate with lance height at different lance angles. At all lance angles, splashing rate increased with increasing lance height from the liquid bath up to a critical distance and then started to decrease with further increase in lance height. When the nozzle was close to the surface, a very deep depression was formed in the liquid bath and the liquid droplets, generated from the edge of the cavity, recirculate into the cavity. With the increase of lance height, the cavity shape slowly converted from penetrating regime to splashing regime and the splashing rate increased. In other words, at higher lance height, the cavity was wider, providing large contact area for increased momentum transfer from gas to liquid. As a result more droplets were generated. When the critical lance height was reached, further increase in lance height decreased the momentum intensity at the surface and splashing rate was reduced.
In case of top blowing, similar trend was observed by the previous researchers [6, 9, 41]. Figure 5-9 also shows that the critical lance height at 35 degree lance angle was higher compared to 40 and 45 degree. This was expected because at 35 degree lance angle, deeper cavity was formed compared to 40 and 45 degree and hence, change of cavity shape from the penetrating mode to splashing mode occurred at higher lance height.

5.4.4 Cavity Oscillations

Figure 5-10(a)-(d) shows the different stages of wave perturbation, observed from the experimental study, at 35 degree lance angle, 0/57Nm³/min flow rates and 65mm lance height. In vertical jetting, the cavity oscillation is more unstable and random because of the lack of a horizontal component. In angled jetting, the vertical component of the jet creates the depression and the horizontal component directs the movement of the cavity horizontally [5]. The experimental studies, performed by the previous researchers [37, 38], also showed similar behaviour of wave perturbation in case of inclined jetting.
Figure 5-10(a) shows the first stage of the cavity formed by angled jetting without any perturbation. As the blowing continued, a small wave perturbation was formed from the left corner of the cavity as shown in Figure 5-10(b). The small wave perturbation was then continued moving away from the lance and started growing bigger. When the
Figure 5-10 Wave progression in angled jetting from CFD and experimental study.

Surface wave reached middle of the cavity, it looked like two different cavities of same size on both sides of the wave as shown in Figure 5-10(c). In Figure 5-10(d), the surface wave moved towards the right corner of the cavity from where it eventually reached close to the water surface and dispersed into splash. This was one complete cycle of
cavity oscillation during inclined jetting. After that another new wave perturbation started forming and the entire process was repeated. Peaslee and Robertson [38] reported that this process is repeated three to nine times per second depending on the velocity of the jet and lance height. In the present experimental study, the frequency of cavity oscillation was found to be around 5 per second which is in the range of reported frequency of oscillations. It was also found from the experimental study that the frequency of oscillation decreased to 4Hz and 3Hz at 105mm and 165mm lance heights respectively. This is in agreement with the previous experimental study [37, 38] where it was reported that the frequency of oscillations decreases with increasing lance heights.

5.4.5 Splashing Distribution on The Wall

The volume of splashed water in each plastic container is plotted in Figure 5-11 against the container position to obtain the splash distribution on the wall for two different lance heights. At lower lance height (65mm), a higher proportion of liquid splashed towards the lower part the converter, close to the bath surface. Splashing rate on the
upper part of the wall was very low compared to the lower part. Again the reason is the shape of the cavity formed at this particular operating condition. Also at lower lance height when the angle of inclination from the vertical was increased, the splashing rate also increased dramatically at the lower part of the wall due to transformation from penetrating to splashing mode. When the lance height was increased, more and more liquid splashed to the upper part of the wall along with the lower part. At 165mm lance height, splashing rate on the upper part of the wall was higher than that of 65mm lance height. This was because at 165mm lance height, the cavity was wider compared to that at 65mm lance height and hence, more of the generated droplets splashed to the upper part of the converter wall. From Figure 5-11, it can be concluded that the wall splashing rate can be reduced by operating at low lance height (penetrating mode), but the lance angle should be selected carefully.
5.4.6 Dimensionless Relationship of Depth of Penetration

The interaction of a vertical gas jet with the liquid was first analysed theoretically by Banks and Chandrasekhara [35] using stagnation pressure analysis method. In the present study, their [35] stagnation pressure analysis has been modified for the inclined jetting case. Figure 5-12 shows the schematic of inclined jetting on a liquid surface:

![Schematic of inclined jetting](image)

Figure 5-12 Schematic of inclined jetting.

The axial centreline velocity \( U_j \) for a turbulent jet can be expressed as:

\[
U_j = \frac{U_0 K_2 d_e}{\gamma} \quad \text{when } Y > K_2 d_e
\]

Where \( U_0 \) is the gas velocity at nozzle exit and \( K_2 \) is a constant. Neglecting the effect of surface tension, the vertical component of the gas dynamic pressure in the proximity of the stagnation point is balanced by the hydrostatic head:
\[ \frac{1}{2} \rho_g U_j^2 \cos \theta = \rho_l p g \]  

Where \( \rho_g \) and \( \rho_l \) is the gas and liquid density \((kg/m^3)\) respectively, \( p \) is the depth of penetration and \( g \) is acceleration due to gravity \((ms^{-2})\). Combining equation (5.2) and (5.3)

\[ \frac{1}{2} \rho_g U_0^2 K_2^2 \frac{d^2}{Y^2} \cos \theta = \rho_l p g \]

\[ \frac{\pi}{4} \rho_g U_0^2 d_2^2 \cos \theta = \frac{\pi}{2K_2^2} \rho_l p g Y^2 \]

\[ \frac{M \cos \theta}{\rho_l p g} = \frac{\pi}{2K_2^2} Y^2 \]

\[ M = \frac{\pi}{4} \rho_g U_0^2 \frac{d_2^2}{Y^2} \]

Now using \( Y = h' + \frac{p}{\cos \theta} \) in equation (5.4) and rearranging we get,

\[ \frac{M \cos \theta}{\rho_l g h'} = \frac{\pi}{2K_2^2} \left( \frac{p}{h'} \right) \left( \frac{p}{h' \cos \theta} + 1 \right) \]

Figure 5-13 shows the plot of dimensionless jet momentum with dimensionless depth. The plot shows a straight line and the slope of this straight line is 34.92 which is equal to \( \frac{2K_2^2}{\pi} \). Hence, the value of \( K_2 \) in the present study is calculated as 7.4, which is in the range of the \( K_2 \) values (6.4 to 7.9) calculated by the previous researchers [35, 41, 42, 51].
Critical Depth of Penetration

In the present study, the critical depth of penetration of water at different lance angle was measured at constant lance height. Critical depth of penetration means the depth at which the droplet generation starts. Chatterjee and Bradshaw [51] proposed an equation to calculate the critical depth of penetration in different liquids for top jetting. They concluded that critical depth of penetration depends only on liquid properties and the influence of lance height is negligible. Shrivastava et al. [116] found that the critical depth of penetration for the onset of splashing decreases with the increasing lance angle from the vertical. Similar observation was made in the present study. Although the penetration depth deceases linearly with lance angle [53], Figure 5-14 shows that the critical depth of penetration does not decrease linearly with lance angle. The critical depth of penetration was found to have a parabolic relationship with the lance angle and can be expressed as:

$$Y = 34.91551 X + 0.1003$$

$$R^2 = 0.9663$$
Using the above equation, the critical depth of penetration at different lance angle can be determined. At zero degree (top jetting) lance angle, critical depth of penetration for water becomes 0.0151 m which is close to the value of 0.0154 m for top jetting case [50]. Further experimental study is required, using liquids with different property, to check the applicability of equation (5.6) for calculating the critical depth of penetration of different fluids.

Figure 5-14 Variation of critical depth of penetration with angle of inclination.

Li and Harris [50] derived an equation for calculating the critical tangential gas velocity at the onset of splashing based on Kelvin-Helmholtz instability:

\[
\frac{1}{2} \rho_g U_g^2 = \sqrt{\sigma g \rho_l}
\]
Where \( U_g \) is the critical tangential gas velocity (\( m/s \)) and is shown in Figure 5-12. They have related the tangential gas velocity \( U_g \) with the free turbulent jet axial velocity \( U_j \) using the following equation:

\[
U_g = \eta U_j \tag{5-8}
\]

Where \( \eta \) defines the fraction of the jet velocity at the impingement point that is converted to the tangential velocity across the gas-liquid interface. From the previous works with top jetting, the value of \( \eta \) was found to be 0.42-0.46 [50]. It is apparent from Figure 5-12 that the value of \( \eta \) will depend on the angle of inclination. At a higher inclination angle from the vertical, the tangential velocity increases. Now combining equation (5.7) in equation (5.8) we get,

\[
\frac{1}{2} \rho_g U_g^2 = \frac{1}{\eta^2} \sqrt{\sigma \rho_l} \tag{5-9}
\]

Where \( \sigma \) is the surface tension (\( N/m \)) of the liquid. Considering the balance of vertical component of gas dynamic pressure and hydrostatic head at critical depth of penetration:

\[
\frac{1}{2} \rho_g U_j^2 \cos \theta = \rho_l g p_c \tag{5-10}
\]

From equation (5.9) and (5.10),

\[
p_c = \frac{\cos \theta}{\eta^2} \sqrt{\frac{\sigma}{\rho_\theta \rho_l}} \tag{5-11}
\]
Equation (5.11) clearly shows that the critical depth of penetration depends on both angle and \( \eta \) value. That is why the critical depth of penetration is not a linear function of angle. From the experimentally determined critical depth of penetration, the value of \( \eta \) can be calculated at different lance angle using equation (5.11). Figure 5-15 shows the variation of \( \eta \) with lance angle. From the experimental data, a relationship can be obtained between the value of \( \eta \) and lance angle:

\[
\eta = \frac{0.4231}{\cos^{0.523}\theta} \approx \frac{0.4231}{\sqrt{\cos\theta}}
\]

Hence, the value of \( \eta \) at different lance angle can be calculated by dividing the top jetting value of \( \eta \) by the factor \( \sqrt{\cos\theta} \).

Figure 5-15 Variation of \( \eta \) with lance angle.
5.4.8 Effect Of Lance Angle on Blowing Number

In order to quantify the droplet generation rate, Subagyo et al.[7] defined a dimensionless Blowing number \( N_B \) which was mentioned in equation (2.32) and is also presented below:

\[
N_B = \frac{\eta^2 \rho_d U_j^2}{2 \sqrt{\sigma g \rho_l}}
\]

Blowing number is an estimate of how many times the critical Kelvin-Helmholtz interface instability has been exceeded. Droplet generation starts when the Blowing number is equal to one. Calculating the value of \( \eta \) using equation (5.12), the critical impact velocities \( (U_j) \) at the onset of splashing at different lance angles were calculated using the above equation assuming Blowing number equal to one and are presented at Table 5-3. The table shows that in case of top jetting, the critical velocity for the onset of splashing is 15.74m/s which was also shown by previous researchers [36]. Table 5-3 also shows that the higher the angle of inclination from the vertical, the lower the critical impact velocity for the onset of splashing. In other words, Blowing number at the gas-liquid interface reaches unity at lower impact velocity because of higher \( \eta \) value.

However, if the lance angle is greater than 79 degree, the value of \( \eta \) exceeds 1 according to equation (5.12) which is theoretically impossible. Also when the lance angle becomes 90 degree, the critical dynamic pressure becomes zero. But there will be some splashing even if the gas jet flows parallel to the interface. So the equation (5.12) should not be used for lance angles higher than 79 degree from the vertical. In EAF steelmaking, the lance angle does not exceed 50 degree from the vertical. Hence, the equations (5.12) can be used for usual EAF steelmaking range. But further experimental study is required to develop a more rigorous equation for calculating \( \eta \) value at different lance angles.
Table 5-3 Critical impact velocities at different lance angles.

<table>
<thead>
<tr>
<th>Angle, θ</th>
<th>η</th>
<th>Critical impact velocity $U_j$ (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.4231</td>
<td>15.74</td>
</tr>
<tr>
<td>5</td>
<td>0.4240</td>
<td>15.71</td>
</tr>
<tr>
<td>10</td>
<td>0.4264</td>
<td>15.62</td>
</tr>
<tr>
<td>15</td>
<td>0.4305</td>
<td>15.47</td>
</tr>
<tr>
<td>20</td>
<td>0.4365</td>
<td>15.26</td>
</tr>
<tr>
<td>25</td>
<td>0.4444</td>
<td>14.98</td>
</tr>
<tr>
<td>30</td>
<td>0.4546</td>
<td>14.65</td>
</tr>
<tr>
<td>35</td>
<td>0.4675</td>
<td>14.24</td>
</tr>
<tr>
<td>40</td>
<td>0.4834</td>
<td>13.78</td>
</tr>
<tr>
<td>45</td>
<td>0.5031</td>
<td>13.23</td>
</tr>
<tr>
<td>50</td>
<td>0.5277</td>
<td>12.62</td>
</tr>
</tbody>
</table>

Subagyo et al.[7] reported that the droplet generation rate increases with Blowing number. The present study shows that at constant lance height and flow rate, the Blowing number on the jet-liquid interface increases with increasing lance angle from the vertical due to an increase in η value. Figure 5-16 shows the variation of Blowing number with lance angle at three different lance heights and 0.43Nm$^3$/min flow rate. The velocity of the gas jet at the interface were calculated by using the following expression [6]:

$$U_j = U_0 \frac{0.97}{0.14H^0.29 + 0.29}$$

5-13
The axial velocity at a distance of less than five nozzle exit diameters is assumed as equal to the nozzle exit velocity which is a reasonable assumption because it was reported that the nozzle exit velocity remain constant for three to eight nozzle exit diameter [3]. It was discussed in the previous section that the wall splashing rate increases with increasing lance angle. But an increase in Blowing number does not always mean an increase in wall splashing rate. The Blowing number during inclined jetting depends on both flow rates and lance angle. If the flow rate is high but the cavity is in penetrating mode, the wall splashing rate may be lower than those operating at low flow rate and in splashing mode. Figure 5-17 shows that at 0.57Nm$^3$/min flow rate and 45 degree lance angle, the Blowing number is $N_B = 22$ which is lower than the Blowing number ($N_B = 29$) at 0.71Nm$^3$/min flow rate and 35 degree lance angle. But the wall splashing rate for the former condition is higher than the later one. The reason behind this is the shape of the cavity. At 0.57Nm$^3$/min flow rate and 45 degree lance angle the cavity is operating in splashing mode whereas at 0.71Nm$^3$/min flow rate and 35 degree lance angle, the cavity is operating in penetrating regime. But, if there is no regime change occurs, an increase in Blowing number will always increase the wall splashing rate as shown in Figure 5-18, where the cavity is operating in splashing regime. Hence,
it is very difficult to come up with an empirical equation for calculating the wall splashing rate at different Blowing number because of the regime change.

Figure 5-17 Wall splashing rate at different blowing number at 65 mm and lance height.

Figure 5-18 Wall splashing rate at different blowing number at 165 mm lance height.
5.4.9 Modelling of Full-Scale Furnace

Achieving complete similarity of a complex system like EAF steelmaking process is difficult. In the present study, a different approach was used to model the gas jet impinging phenomenon in the real process using the room temperature water model. But no effort was made to model the magnetic force resulting from the electrode. In order to maintain the dynamic similarity between the real process and water modelling, previous researchers have used the dimensionless modified Froude number [53]:

\[ Fr_m = \frac{\rho g U_0^2}{\rho g D} \]

Where \( U_0 \) is the gas jet velocity (m/s) at the nozzle exit and \( D \) is the characteristic dimension (m). Using the modified Froude number, the velocity of the gas jet at the nozzle exit in case of water model was calculated in the present study. Then the Blowing number similarity was maintained at the liquid surface because it was reported by Subagyo et al. [7] that similar Blowing number at the jet-liquid surface results in similar droplet generation rate. The present authors developed a CFD model to calculate the supersonic jet velocity characteristics at steelmaking temperature (chapter 3). Using this model and equation (2.32), the Blowing number at the liquid surface inside the real furnace was calculated for a particular lance angle. Then, from the Blowing number similarity, the velocity of the gas jet on the water surface \( U_j \) required to maintain the similar Blowing number in water model was calculated. After that, the lance height (\( h' \) in Figure 5-12) required to decay from nozzle exit velocity (\( U_0 \)) to the water surface velocity (\( U_j \)), was determined using equation (5.13) [6]. Then the operating condition in the experiment was fixed to that particular gas velocity and lance height and the splashing rates were measured. The dimensional analysis for the modelling of real furnace of OneSteel operating in Laverton is shown in

Table 5-4:

Table 5-4 Dimensional analysis of the full-scale furnace.
<table>
<thead>
<tr>
<th>Parameters</th>
<th>One Steel</th>
<th>One-fifth scale model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Liquid/gas</td>
<td>Steel/Oxygen</td>
<td>Water/air</td>
</tr>
<tr>
<td>Liquid density</td>
<td>$7000 \text{kg/m}^3$</td>
<td>$1000 \text{kg/m}^3$</td>
</tr>
<tr>
<td>Gas density at nozzle exit</td>
<td>$2.3 \text{kg/m}^3$</td>
<td>$1.18 \text{kg/m}^3$</td>
</tr>
<tr>
<td>Volumetric flow rate</td>
<td>$40 \text{Nm}^3/\text{min}$</td>
<td>$0.1984 \text{Nm}^3/\text{min}$</td>
</tr>
<tr>
<td>Lance diameter</td>
<td>$31.75 \text{mm}$</td>
<td>$6 \text{mm}$</td>
</tr>
<tr>
<td>Velocity at nozzle exit</td>
<td>$490 \text{m/s}$</td>
<td>$111 \text{m/s}$</td>
</tr>
<tr>
<td>Modified Froude number</td>
<td>$249.166$</td>
<td>$249.166$</td>
</tr>
<tr>
<td>Lance height</td>
<td>$1.5 \text{m}$</td>
<td>$0.046 \text{m}$</td>
</tr>
<tr>
<td>Velocity on liquid surface</td>
<td>$79 \text{m/s}$</td>
<td>$79 \text{m/s}$</td>
</tr>
<tr>
<td>Blowing number</td>
<td>$35.63$</td>
<td>$35.63$</td>
</tr>
<tr>
<td>Lance angle</td>
<td>$45 \text{ degree}$</td>
<td>$45 \text{ degree}$</td>
</tr>
</tbody>
</table>

The splashed liquids were collected at 65 nozzle exit diameter ($65 \times 6 = 390 \text{mm}$) from the jet impingement point in the water model and the splashing rates were calculated. The splashing rate at 45 degree lance angle was calculated as $0.27 \text{ml/s}$ which is equal to $0.27 \text{g/s}$ for water. As the Blowing number similarity were maintained here, it can be said that the splashing rates will be the same in case of real furnace assuming only steel as liquid with no slag on the surface. In case of real furnace the forward splashing rate of steel are expected to be $1.91 \text{g/s}$ (weight of splashed steel). However, industrial trials are required to confirm this prediction.

The present experimental study was performed using a thin slice model to get a better view of the cavity shape. The depth of penetration in the 3D model for a particular jet velocity would be slightly lower than that in thin slice model because jet spreads in two directions only in thin slice model. Hence, the value of $K_2$ in equation (5.5) would be slightly different in case of 3D model. Thin slice nature of the model may have some effect on the critical penetration depth measurement also. But the present experimental study gives a better understanding of the splashing phenomenon in the case of inclined jetting.
In the present study, the no-slag condition was considered. We assumed that the impinging gas jet pushes the slag layer away from the impingement zone and exposes the liquid metal to the high speed gas jet [42]. Previous studies [7, 38] suggested that the metal droplets are generated from the gas-metal interface and then the droplets may splash outside or recirculate inside the cavity depending on the impingement mode. If a slag layer is present on top of the liquid metal, some metal droplets with low kinetic energy will end up in the slag layer and reduce the wall splashing rate. Previous studies [5, 53, 117] showed that thicker, more viscous and foamy slags result in less splashing.

5.5 Conclusions

Water modelling experiments have been carried out to understand the effect of different operating parameters (lance angle, lance height and flow rate) on depth of penetration and wall splashing rate in steelmaking. The following conclusions can be drawn from this study:

1. The forward splashing rate increases with the increase of lance angle and flow rate.

2. The splashing rate is not a monotonous function of lance height. Splashing rate increases with lance height up to a critical distance and after that splashing rate decreases with the increase of lance height. The trend is similar at all lance angles from 35 to 45 degree.

3. The splashing rate is lowest when operating in penetrating regime.

4. The splashing rate is highest at the lower region of the wall, close to liquid surface.

5. The use of higher lance height (less than the critical lance height) increases the overall splashing rate and more liquid splashes on the upper part of the wall compared to low lance height. At low lance height, splashing rate is reduced and is concentrated on the lower part of the wall.
6. For inclined jetting, droplet generation starts at shallower critical depth of penetration and at lower impact velocity. Critical depth of penetration is proportional to the square of the cosine of lance angle.

7. The Blowing number at the jet-liquid interface increases with the increase of lance angle from the vertical. An increase in Blowing number doesn’t always increase the droplet generation rate.

According to the present experimental investigation, wall splashing rate can be reduced by placing the nozzle close to the surface and operating in penetrating regime. But there is a chance of nozzle wear due to the back splashing of liquid droplets on the nozzle. This can be avoided by the introduction of coherent jet nozzle [25] in the EAF steelmaking process. A coherent jet gives the advantage of installing the nozzle at higher lance height because it can maintain the higher impact pressure on the bath surface to produce the penetrating regime even at higher lance height as discussed in chapter 3 of this thesis. But the angle of inclination should be carefully selected because it is shown in the present study that slight change in angle increases the splashing rate dramatically because of change in impinging mode from penetrating to splashing.
Chapter 6

6.1 Introduction

This chapter describes the numerical modelling of inclined jetting on liquid surface as used in electric arc furnace (EAF) steelmaking process. The effect of lance angles and gas flow rates on the depth of penetration are numerically investigated and validated against the experimental data. The possible reasons for the discrepancy between numerical and experimental results have been described. The shape of the cavity formed, and the subsequent movement have been discussed from the numerical model and also compared against the experimental observations. Finally, effort has been made to quantify the droplet generation rate from the numerical model and the difficulties involved in the simulation of splashing have been outlined.

6.2 Governing Equations

In the present study, the volume of fluid method [68] (VOF) was used to simulate the sharp gas liquid interface. The governing equations for solving the gas-liquid interaction using the VOF method was described in detail in section 4.2.3 and therefore, not repeated here. The modelling of turbulence was carried out using the $k – \varepsilon$ turbulence model [14] which was discussed in section 4.2.4.

6.3 Computational Domain

The computational domain with the boundary conditions used in the present numerical model has been presented in Figure 6-1. The height, length and width of the computational domain are 1000mm, 1500mm and 12.5mm respectively. The dimensions of the computational domain were selected according to the experimental study described in Chapter 5 with some modifications which are presented below:

- The bottom floor of the computational domain is flat whereas the bottom floor of the experimental study was round. But the height of the liquid bath in the experimental study was kept high enough to ensure that the cavity oscillation was not influenced by the bottom surface.
• The width of the experimental set up was 25mm. But in the computational domain, the width was taken as 12.5mm and a symmetry boundary condition was used along the plane of the jet centreline. This approach will greatly reduce computational time.

• In the experimental study, the splashed liquid was collected by inserting small plastic containers attached to an aluminium rod at a certain position. In the computational domain, the region behind the splash collectors was not considered as it will have negligible effect on the simulation and also will save computational time due to the reduced number of cells. The small plastic containers were termed as splash collector and were numbered from one to eleven.

Figure 6-1 Computational domain with boundary conditions.
The inlet face of the computational domain was inclined at a certain angle from the vertical and the inclination angle was set according to the experimental operating conditions.

### 6.4 Boundary Conditions

The boundary conditions were selected according to the experimental study. Normal velocity boundary condition was used at the inlet surface. The nozzle exit velocities were calculated from the experimental flow rates data. The boundary values of the turbulence quantities at the inlet face were imposed according to equation (4.16). Atmospheric pressure boundary conditions were used at the outlet and also at the splash collecting faces. At the symmetric side (Figure 6-1) of the computational domain, symmetry boundary condition was used. The no slip boundary condition for the velocities was used at the solid impermeable and smooth wall. In order to model the near-wall effects (e.g. viscous damping) close to the wall, the compound wall treatment (CWT) (known as hybrid wall function in AVL Fire) approach [114] was used which was described in details in section 4.2.6. Table 6-1 shows the lance height, lance angles and nozzle exit velocities used in the present simulation.

Table 6-1 Lance height, lance angle and the nozzle exit velocity in the present simulation.

<table>
<thead>
<tr>
<th>Lance height (mm)</th>
<th>Lance angle (Degree)</th>
<th>Flow rates (Nm³/min)</th>
<th>Nozzle exit velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>65</td>
<td>35</td>
<td>0.71</td>
<td>76</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>61.7</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.43</td>
<td>46.5</td>
</tr>
<tr>
<td>45</td>
<td>0.71</td>
<td>76</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.57</td>
<td>61.7</td>
<td></td>
</tr>
<tr>
<td></td>
<td>0.43</td>
<td>46.5</td>
<td></td>
</tr>
</tbody>
</table>

The physical properties of the two fluids used in the present numerical simulation are presented in Table 6-2.
Table 6-2 Physical properties of fluids at 303K temperature.

<table>
<thead>
<tr>
<th>Properties</th>
<th>Air</th>
<th>Water</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m$^3$)</td>
<td>1.18</td>
<td>995</td>
</tr>
<tr>
<td>Viscosity (Ns/m$^2$)</td>
<td>$1.824 \times 10^{-5}$</td>
<td>0.000798</td>
</tr>
<tr>
<td>Surface tension (N/m)</td>
<td>-</td>
<td>0.071</td>
</tr>
</tbody>
</table>

6.5 Computational Procedure

The computational model was developed using the finite volume based commercial CFD software AVL Fire Version 2009.2 and the computations were carried out in an Intel Xeon work station with 2.67GHz quad core processor and 8GB of Ram. The unsteady, incompressible, continuity, momentum and energy equations were solved using segregated solver to calculate the pressure and the velocity field. Equation (4.3) was solved first at the beginning of each time step. The volume fraction field is then used to compute the mixture viscosity and density using equations (4.8) and (4.9). Then the incompressible continuity (4.4) and momentum (4.5) equations were solved using SIMPLE algorithm [94] which executes a sequence of predictor-corrector steps utilizing a pressure equation that is derived from a combination of equation (4.4) and (4.5). For the discretization of momentum and pressure correction equation, the AVL SMART scheme [92] and central differencing scheme (CDS) was used. The simulations were run for 10s to guarantee the stability in the solution although the simulations reached the quasi steady state before that time. Stable transient solutions were obtained with the time step size of 0.0005s to 0.00001s. The total number of cells in the computational domain was 386443. A grid independency test was carried out to analyse the sensitivity of the solution to the grid size.

6.6 Results and Discussions

6.6.1 Gas Jet Penetration

Figure 6-2(a) and (b) shows the effect of varying gas flow rates (0.71Nm$^3$/min, 0.57Nm$^3$/min and 0.43Nm$^3$/min) on the depth of penetration (dimension “p” in Figure 2-13) at 35$^0$ and 45$^0$ lance angles. The lance height was kept constant at 65mm in all
Figure 6-2 Variation of depth of penetration with flow rates at (a) 35 degree and (b) 45 degree lance angle.
cases. Both the figures show that the depth of penetration increases with increasing gas flow rates. It is also evident from the figures that the depth of penetration decreases with increasing lance angle from the vertical which are expected because the forces in the vertical direction are directly proportional to the cosine of the jet angle. The CFD model unpredicted the depth of penetration at all lance gas flow rates and lance heights. The average percentage of errors between the predicted and measured depth of penetration were 20% and 25% respectively for 45 and 35 degree lance angles respectively. These differences might be the result of numerical errors involved in the simulation which may arise from the discretization of the partial differential equation and also from the differencing scheme used in the simulation. Fine grid was used to reduce discretization error in the simulation. Another reason might be that the CSF model [74] which was used in the present study to model the momentum interfacial exchange is based on the simplified boundary conditions. Hwang and Irons [75] showed that the full stress boundary conditions at the gas liquid interface can predict the waves and cavity depths on the surface more accurately.

Figure 6-3(a) and (b) shows that the diameter of depression along the direction of the jet (dimension “b” in Figure 2-13) also increases with increasing lance angle. The figures
Figure 6-3 Variation of diameter of depression with flow rates at (a) 35 degree and (b) 45 degree.

also show that, unlike depth of penetration, the diameter of depression increases with increasing lance angle from the vertical at all flow rates. This occurs because the horizontal component of the jet increases with increasing lance angle from the vertical. There is good agreement between the predicted and measured diameters of the cavity as shown in Figure 6-3. With increasing gas flow rates, surface waves were generated inside the cavity which has been described in section 6.6.3.

6.6.2 Grid Sensitivity Test

In order to study the grid sensitivity of the solution, calculations of depth of penetration in case of lower shrouding flow rate was carried out using three different grid levels: coarse grid (191522 cells), medium grid (383443 cells) and fine grid (562593 cells). The calculated depths of penetration for all grid levels are shown in Figure 6-4. The variation in the depth of penetration, calculated with coarse and medium grid, is around 6%, and it becomes less than 2% between medium and fine grid. Hence, it can be said
that the solution is not sensitive to the grid. The solution with fine grid takes longer time than the solution with medium grid. Therefore, considering the computational time, the results obtained with medium grid were used for analysis and discussion in this study.

![Figure 6-4 Calculated depth of penetration at different grid levels.](image)

6.6.3 Cavity Shape and Oscillation

In vertical jetting, the cavity oscillation is more unstable and random because of the lack of a horizontal component. In angled jetting, the vertical component of the jet creates the depression and the horizontal component directs the movement of the cavity horizontally [5].

![Image](image)
Figure 6-5 Wave progression in angled jetting from CFD and experimental study (chapter 5).

Figure 6-5(a)-(d) shows the different stages of wave perturbation inside the cavity, during inclined jetting from both the CFD simulation and experimental study (Chapter
5). The CFD simulation was found to agree well with experimental study. Figure 6-5(a) shows the first stage of the cavity formed by angled jetting without any perturbation. As the blowing continued, a small wave perturbation was formed from the left corner of the cavity as shown in Figure 6-5(b). The small wave perturbation then continued moving away from the lance and started growing bigger. When the surface wave reached middle of the cavity, it looked like two different cavities of same size on both sides of the wave as shown in Figure 6-5(c). In Figure 6-5(d), the surface wave moved towards the right corner of the cavity from where it eventually reached close to the water surface and dispersed into splash. This was one complete cycle of cavity oscillation during inclined jetting. In the present CFD model the frequency of cavity oscillation at 35 degree angle, 0.57Nm$^3$/min and 65mm lance height was found to be around four times per second compared with 5 times per second observed from the experimental study (Chapter 5). One of the reasons for the underprediction of oscillation frequency might be the $k - \varepsilon$ turbulence model used in the simulation of jet liquid interactions. In the present study, turbulence was assumed homogeneous in both the phases. Another reason might be the CSF model which was used in the present study for modelling the surface tension effect. The CSF model is based on the simplified boundary conditions where the gas side shear stress is neglected. Hwang and Irons [75] showed that the full stress boundary conditions at the gas liquid interface can predict the surface waves and cavity dimensions more accurately.

### 6.6.4 Prediction of $\eta$ Value With Lance Angles

The experimental study, presented in chapter 5, showed that the value of $\eta$ in the Blowing number theory increases with the increase in lance inclination angle from the vertical. In order to predict the $\eta$ value using the present CFD model, the turbulent jet velocity ($U_j$) and the critical gas velocity ($U_g$) were defined at the jet liquid interface as shown in Figure 6-6:
The values of $U_j$ and $U_g$ were predicted at three different flow rates and two different lance angles $35^0$ and $45^0$ using CFD model. Then, using equation (2.31), average values of $\eta$ was calculated for the $35^0$ and $45^0$ lance angles.

Figure 6-7 shows the variation of $\eta$ values with lance angles calculated from both the CFD model and empirical equation proposed by the present author in section 5.4.7. The figure shows that CFD model overpredicts the $\eta$ value at both lance angles. The percentage of deviation is 12% and 8% for $45^0$ and $35^0$ lance angles respectively. One of the reasons of this difference might be the grid used in the present study. A finer grid might predict the very fine droplets, generated due to splashing, and also the jet distribution more accurately. But this would be very costly in terms of computational time. Another reason might be the rectangular grid used for inclined jetting where gas jet flows diagonally through the grid. The AVL SMART scheme was used to minimize any error resulting from the cross flow.
6.6.5 Modelling of Wall Splashing Rate

In the present study, effort was made to numerically quantify the wall splashing rate. This was done by adding the mass flow rates of liquid phase through the splash collector boundary faces as shown in Figure 6-1. There were eleven splash collector boundary regions in the computational domain which were positioned according to the experimental study described in chapter 5. The wall splashing rate was calculated by following equation:

\[
\dot{m} = \sum_{k=1}^{11} \left( \sum_{i=1}^{f} (\alpha_i \rho_l A U_l)_i \right)_k
\]

Equation (6-1)

Where \( k = 1, 2, 3 \ldots \ldots 11 \) denotes the number of splash collector regions, \( i = 1, 2, 3 \ldots \ldots f \) is the number of faces in each splash collector region, \( \alpha_i \) is the volume fraction of the liquid phase, \( \rho_l \) is the density of liquid phase, \( A \) is the area of the face and \( U_l \) is the velocity of liquid phase. Equation (6-1) was used to calculate the wall splashing rate.
at different operating conditions (0.71Nm³/min, 0.57Nm³/min and 0.43Nm³/min flow rates, 65mm lance height, 35° and 45° lance angles). The CFD model was found to over-predict the wall splashing rate when compared with the experimental wall splashing rate both at 35° and 45° lance angles. The principle reason for the deviation of CFD results from the experimental measurement is the numerical grid used for the modelling of splashing. The present computational domain was unable to capture fine droplets produced as a result of jet-liquid interaction. From extensive literature review, Dogan [118] reported that the droplet size varies from 0.05mm to 5mm. In order to capture these tiny droplets, very fine computational grid is required. In the present study, the minimum grid size was 5mm×6mm×3mm which is too big to capture the generated fine droplets. As a result, the liquids, ejecting out of the cavity, could not be disintegrated into small droplets and all of them splashed at the bottom region of the wall in the forward direction. If it could have been disintegrated into small droplets, it would have followed the gas stream and forward splashing rate would have been reduced. The present computational domain has 386443 cells and requires around 5 days for a simulation equal to 5s real time. A denser computational grid will further increase the computational time and is beyond the scope of the present study. In order by-pass the grid resolution problem, Lagrangian approach was tried for modelling the droplet generation rate assuming a semi-fixed cavity shape which is presented in Appendix C. The Lagrangian approach produced some reasonably good results but needs further improvement before relying on it for predicting the droplet generation rates.

### 6.7 Conclusions

Numerical modelling of inclined air jetting on water has been carried out at different flow rates and lance angles. The CFD model underpredicts the depth of penetration at all operating conditions although the predicted diameter of the cavity is in good agreement with the experimental data. The possible reasons behind this underprediction have been explained. Also, the CFD model realistically reproduces the cavity shapes, perturbations and movements with reasonable accuracy when compared with the experimental observations. The different stages of the cavity wave progression have been predicted reasonably well. The constant $\eta$, used in the Blowing number
calculation, has been calculated for different lance angle. The $\eta$ value increases with the increase of lance inclination angle from the vertical and is in reasonable agreement with the empirical equation proposed from the experimental study in chapter 5. The modelling of droplet generation rate, using Eulerian-Eulerian VOF approach, is still a big challenge for CFD. The principle limitation in the modelling of droplet generation rate is the grid size of the computational domain. Very small grid size is required to capture the fine droplet which is computationally very expensive.
Chapter 7

7. Conclusions and Recommendations For Future Study
7.1 Conclusions

The fluid dynamics aspects of the steelmaking process have been investigated through several computational fluid dynamics (CFD) model and one experimental study at room ambient temperature using air-water system. From the present study, the following conclusions can be drawn:

- The potential core length of supersonic oxygen jet at steelmaking temperature is approximately 2.5 times higher than that of room ambient temperature. This occurs due to the reduction in the growth rate of turbulent mixing region at higher ambient temperatures and associated lower density of the gases entrained into the jet. The dynamic pressure of the jet, which characterizes the momentum transfer, is also higher at high ambient temperatures but with increasing distance from the nozzle exit plane, the relative difference between the dynamic pressure values at different ambient temperatures decreases. Because, the temperature of the jet increases with increasing distance from the nozzle exit which in turn decreases the density as well as dynamic pressure of the jet. Hence, if the lance height is very high, the impact dynamic pressure will be almost similar at all ambient temperatures.

- The standard $k-\varepsilon$ turbulence model, with compressibility correction, underpredicts the potential core length of the supersonic jet at high ambient temperatures. An ad-hoc modification of the constant $C_\mu$ in the turbulent viscosity calculation has been proposed. This modification takes into account the local temperature gradient and reduces the value of $C_\mu$ at higher ambient temperatures which in turn reduces the turbulent viscosity. Lower turbulent viscosity results in lower turbulent shear stress and hence, reduction in the growth rate of turbulent mixing region leading to increased potential core length. The validity of this model has been confirmed by comparing the velocity and temperature profile obtained by CFD model with the available experimental data.

- The potential core length of the shrouded supersonic jet (also known as coherent jet) is longer than that of conventional supersonic jet. The potential core length also depends on the gas used as the central supersonic jet. The present study
shows that the potential core length of the coherent supersonic oxygen jet is higher than that of the coherent supersonic nitrogen jet. This occurs because the temperature of the combustion flame is higher in case of coherent oxygen jet compared with the coherent nitrogen jet. The higher combustion flame temperature further reduces the ambient density as well as turbulent shear stress and lowers the jet mixing with the surroundings. The reason for higher combustion flame temperature is the secondary combustion that takes place in the shear layer when oxygen is used as the central supersonic jet.

- From the present CFD study, the potential core length of the supersonic coherent oxygen jet is 4.2 times longer than that of the free supersonic oxygen jet at room ambient temperature. At steelmaking temperatures, the potential core length of the coherent supersonic oxygen jet is 1.4 times longer than that at room ambient temperature. Hence, the potential core length of the coherent supersonic oxygen jet is about 5.8 times longer than that of free supersonic oxygen jet at room ambient temperature. Thus, the coherent jet can be installed far from the liquid surface which will reduce the splashing of liquid metal on the lance tip and hence, extend the lance tip life.

- Use of two different computational domains, for modelling the supersonic shrouded jet impingement on liquid surface, improves the prediction of depth of penetration when compared with the previous approach where a single domain is used for modelling the supersonic jet impingement assuming the supersonic jet incompressible.

- The coherent length of the supersonic shrouded jet increases with the increase of shrouding gas flow rates. In this study, shrouding was created using only compressed air. The depth of penetration increases with increasing shrouding flow rates. Also, the higher shrouding gas flow rate increases the velocity of the liquid bath which would result in reduced gas-liquid mixing time inside the furnace.

- The critical tangential gas velocity is constant for a particular jet inlet velocity and is independent of lance height. Therefore, the cavity surface area might be the most important factor in the generation of liquid droplets. The higher the
surface area, the more is the droplets generation rate. The cavity surface area is lower while operating in penetrating mode and then increases when the cavity transforms from the penetrating to splashing mode with increasing lance height. After a certain lance height, the momentum of the jet becomes the dominant factor and the droplet generation rate decreases with the increase of lance height because the jet momentum on the liquid surface decreases.

- An increase in Blowing number does not always increase the droplet generation rate. The Blowing number theory fails if the lance height is low and the cavity operates in penetrating mode.

- The reported $\eta$ value in the literature was calculated at the onset of splashing when the cavity transforms from the dimpling to splashing mode. The present CFD study shows that the $\eta$ value is a function of lance height and a constant $\eta$ value cannot be used at all lance heights. From the present experimental study, it has also been proposed that the $\eta$ value changes with the lance angle according to the following equation:

\[
\eta = \frac{0.4231}{\sqrt{\cos \theta}}
\]

- The Blowing number at the jet-liquid interface increases with the increase of lance angle from the vertical if the axial lance height remains constant.

- The forward splashing rate increases with the increase of lance angle from vertical and with flow rate. The splashing rate is highest at the lower region of the wall, close to liquid surface.

- In case of inclined jetting, droplet generation starts at shallower critical depth of penetration. Critical depth of penetration follows the following relationship with lance angle when water is used as the liquid:

\[
p_c = 0.0151 \cos^2 \theta
\]
• The CFD model predicts the cavity progression and oscillation in case of inclined jetting with reasonable accuracy.

• Modelling of droplet generation, using Eulerian approach, requires very fine mesh which is computationally intensive.

7.2 Contribution to New Knowledge

The following findings are considered to be novel from the present study:

• The $k - \varepsilon$ turbulence model with compressibility correction underpredicts the potential core length of the supersonic jet at higher ambient temperatures. A temperature corrected $k - \varepsilon$ turbulence model has been proposed to take into account the large temperature gradient and reduce the turbulent mixing which in turn results in longer potential core length of the jet.

• This shrouding combustion flame in case of coherent jets reduces the turbulent shear stress magnitude in the shear layer. It also delays the merging of the vorticity region, which is a measure of jet mixing with the surrounding, to the jet centreline. As a result, the potential core length of the jet increases.

• Critical depth of penetration for the onset of splashing decreases with the increase of lance inclination angle which results in an increase in droplet generation rate for inclined jetting. A modification of the Blowing number theory has been proposed by including the effect of lance angle.

• It has been shown that Blowing number theory fails if the lance height is low and the cavity operates in penetrating mode. Hence, cavity mode should be considered as a variable in the Blowing number theory.


### 7.3 Recommendations for Further Study

In this section, several recommendations have been made to improve and extend the present research which, if implemented, would contribute to the better understanding the supersonic jet behaviour and the jet-liquid interactions inside steelmaking furnace. These recommendations are listed below:

- The modification proposed for the $k - \varepsilon$ turbulence model works only for the cold supersonic jet discharging into a hot ambient atmosphere by reducing the growth rate of turbulent mixing layer. Further study is required to develop a comprehensive temperature corrected turbulence which would work both at hot and cold ambient atmosphere by decreasing and increasing the growth rate of turbulent mixing region respectively.

- Only one step complete combustion reaction between CH$_4$ and O$_2$ has been considered in the coherent jet modelling whereas in reality the combustion occurs in a number of steps producing different types of minor species. The predictions can be improved further by incorporating multi-step combustion reactions in the combustion modelling.

- The coherent length of the supersonic coherent jet is a function of shrouding fuel and oxidant flow rates. It would be beneficial for the steelmakers if an optimum condition in terms of longer coherent length can be determined relating the fuel, oxidant and main supersonic jet flow rates. Also, the position of the annular holes might have an effect on the potential core length of the jet. Further study is required to predict the axial jet velocity distributions at different annular inlet hole arrangements and to determine their efficient position.

- An experimental investigation on the variation of $\eta$ value with lance heights can provide quantitative validation of the reported $\eta$ values in the present CFD model. A relationship between the $\eta$ value, cavity dimensions, lance angle and lance height is required for predicting the droplets generation rate more accurately.

- The present CFD model shows that the critical gas velocity ($U_g$) is independent of lance heights. It would be interesting to see from the experimental study that
how the extra kinetic energy of the jet at lower lance height is used for holding the cavity and stirring the bath.

- The CFD model of jet impingement on the liquid surface can be extended to include the chemical reactions and heat transfer at steelmaking conditions. The results would be helpful to understand the impact zone chemistry.
References
World steel in Figures. 2011, World steel association.


Appendix A

Error analysis in the splashing rate measurement

If \(a\) is volume of splashed liquid and \(b\) is the time of splash collection then the splashing rate \(p\) can be calculated using \(p = \frac{a}{b}\). The error in the calculation of \(p\) is:

\[
\frac{\delta p}{p} = \frac{\delta a}{a} + \frac{\delta b}{b}
\]

Where \(\delta a\), \(\delta b\) and \(\delta p\) are the error in \(a\), \(b\) and \(p\) respectively.

For example, at 45 degree lance angle and 65mm lance height,

Time of splash collection= 190.5sec

Volume of splashed liquid= 246.8ml

Splashing rate=1.296ml/sec.

Hence, the error in the measurement of splashing rate:

\[
\delta p = p \left( \frac{\delta a}{a} + \frac{\delta b}{b} \right) = 1.296 \left( \frac{0.2}{246.8} + \frac{0.05}{190.5} \right) = 0.0014 \text{ ml/sec}
\]

The following table shows the calculated error in the splashing rate measurements at all operating conditions. The maximum error was found to be 0.008ml/sec in case of 45 degree lance angle and 105mm lance height.
Table 1A: Errors in splashing rates calculation.

<table>
<thead>
<tr>
<th>Lance heights (m)</th>
<th>Angles (degree)</th>
<th>Flow rates (Nm$^3$/min)</th>
<th>Error (ml/sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.065</td>
<td>35</td>
<td>0.43</td>
<td>0.00018</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.0003</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>0.00032</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.43</td>
<td>0.00029</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.00066</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>0.0014</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.43</td>
<td>0.0004</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.001</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>0.0046</td>
</tr>
<tr>
<td>0.105</td>
<td>35</td>
<td>0.43</td>
<td>0.0002</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.0007</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>0.0014</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.43</td>
<td>0.0003</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.00084</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>0.0023</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.43</td>
<td>0.0004</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.00136</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>0.008</td>
</tr>
<tr>
<td>0.125</td>
<td>35</td>
<td>0.57</td>
<td>0.00077</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.57</td>
<td>0.0012</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.57</td>
<td>0.0021</td>
</tr>
<tr>
<td>0.145</td>
<td>35</td>
<td>0.57</td>
<td>0.00086</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.57</td>
<td>0.001</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.57</td>
<td>0.002</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>-------</td>
<td>-----</td>
<td>-----</td>
<td>-------</td>
</tr>
<tr>
<td>0.165</td>
<td>35</td>
<td>0.43</td>
<td>0.0002</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.0007</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>0.0023</td>
</tr>
<tr>
<td>40</td>
<td></td>
<td>0.43</td>
<td>0.0003</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.00088</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>0.00368</td>
</tr>
<tr>
<td>45</td>
<td></td>
<td>0.43</td>
<td>0.0006</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.00154</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>0.0052</td>
</tr>
</tbody>
</table>
## Appendix B

### Experimental Data

Table 1B: Depths and diameters of depression at different operating conditions.

<table>
<thead>
<tr>
<th>Lance heights (m)</th>
<th>Angles (degree)</th>
<th>Flow rates (Nm$^3$/min)</th>
<th>Depth (mm)</th>
<th>Diameter (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.065</td>
<td>35</td>
<td>0.43</td>
<td>53.48</td>
<td>118.96</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>66.93</td>
<td>141.55</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>91.7</td>
<td>183</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.43</td>
<td>52.21</td>
<td>128.43</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>65.9</td>
<td>158.8</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>88.5</td>
<td>189.6</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.43</td>
<td>49.4</td>
<td>136</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>61.96</td>
<td>163.33</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>76.96</td>
<td>200</td>
</tr>
<tr>
<td>0.105</td>
<td>35</td>
<td>0.43</td>
<td>42.33</td>
<td>116</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>59.86</td>
<td>139.65</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>78.5</td>
<td>188.5</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.43</td>
<td>41.8</td>
<td>128.65</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>56.36</td>
<td>161.07</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>78.5</td>
<td>190.7</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.43</td>
<td>40.08</td>
<td>137.4</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>55.5</td>
<td>167</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>69.4</td>
<td>204.5</td>
</tr>
<tr>
<td></td>
<td>35</td>
<td>0.57</td>
<td>58.32</td>
<td>149.5</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
<td>------</td>
<td>-------</td>
<td>-------</td>
</tr>
<tr>
<td>0.125</td>
<td>40</td>
<td>0.57</td>
<td>50.5</td>
<td>159.3</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.57</td>
<td>50</td>
<td>166</td>
</tr>
<tr>
<td>0.145</td>
<td>35</td>
<td>0.57</td>
<td>53.7</td>
<td>143.4</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.57</td>
<td>51.78</td>
<td>154.26</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.57</td>
<td>46.96</td>
<td>168.93</td>
</tr>
<tr>
<td>0.165</td>
<td>35</td>
<td>0.43</td>
<td>33.4</td>
<td>109.65</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>50.72</td>
<td>147.93</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>75.48</td>
<td>201.72</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.43</td>
<td>32.3</td>
<td>124.62</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>49.53</td>
<td>162.32</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>72</td>
<td>197.14</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.43</td>
<td>31.83</td>
<td>128.86</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>48.27</td>
<td>172.88</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>62.87</td>
<td>199.84</td>
</tr>
</tbody>
</table>
Table 2B: Splashing rates at different operating conditions.

<table>
<thead>
<tr>
<th>Lance heights (m)</th>
<th>Angles (degree)</th>
<th>Flow rates (Nm$^3$/min)</th>
<th>Splashing rates (ml/sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.065</td>
<td>35</td>
<td>0.43</td>
<td>0.1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.34</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>0.41</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.43</td>
<td>0.2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>0.785</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>1.295</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.43</td>
<td>0.243</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>1.09</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>3.51</td>
</tr>
<tr>
<td>0.105</td>
<td>35</td>
<td>0.43</td>
<td>0.32</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>1.02</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>1.55</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.43</td>
<td>0.39</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>1.2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>2.4</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.43</td>
<td>0.514</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>1.89</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>6.2</td>
</tr>
<tr>
<td>0.125</td>
<td>35</td>
<td>0.57</td>
<td>1.15</td>
</tr>
<tr>
<td>40</td>
<td></td>
<td>0.57</td>
<td>1.55</td>
</tr>
<tr>
<td>45</td>
<td></td>
<td>0.57</td>
<td>2.45</td>
</tr>
<tr>
<td>0.145</td>
<td>35</td>
<td>0.57</td>
<td>1.25</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.57</td>
<td>1.35</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.57</td>
<td>2.23</td>
</tr>
<tr>
<td>0.165</td>
<td>35</td>
<td>0.43</td>
<td>0.25</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>1.1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>2.883</td>
</tr>
<tr>
<td></td>
<td>40</td>
<td>0.43</td>
<td>0.35</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>1.25</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>3.42</td>
</tr>
<tr>
<td></td>
<td>45</td>
<td>0.43</td>
<td>0.475</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.57</td>
<td>1.81</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.71</td>
<td>5.02</td>
</tr>
</tbody>
</table>
Appendix C

Lagrangian approach of calculating wall splashing rate

In order to solve the grid resolution problem associated with the Eulerian approach, the droplets were tracked with the Lagrangian approach through the computational grid used for solving the gas phase partial differential equations. This approach was used by the previous researchers for tracking individual particle in different industrial process [119, 120]. In the present study, full two-way coupling (interaction) between the gas and liquid phases was taken into account.

Figure C-1 shows the computational domain used with the Lagrangian approach [121] for simulating splash generation. Although it was shown in section 6.6.3 that the cavity and free liquid surface changes constantly, a semi fixed cavity shape was assumed for Lagrangian model as shown in Figure C-1. This was performed by averaging the cavity.

Figure C-1 Computational domain for Lagrangian approach of simulating splashing.
measurement at different time steps from the Eulerian CFD prediction for the corresponding lance height, lance angle and flow rate. Figure C-1 shows the average cavity shape for 65mm lance height, 45° lance angle and 0.57Nm³/min flow rates. Water was introduced through the base of the cavity in the form of wallfilm. The rate of wall film inlet was determined through trial and error by continuously increasing the flow rate until there is no change in the wall splashing rate results. This was done to make sure that the cavity surface always stays wet and had sufficient supply of water as is in the real-life experimental set-up. In this way, it represents the cavity formed on a water surface. The cavity surface was oscillated at a certain frequency, determined from the experimental study, to represent the cavity oscillations described section 6.6.3. In the present study, an oscillation frequency of 6Hz was used.

Air was injected through the air inlet face as shown in Figure C-1. The air jet then impinges on the cavity surface and generates droplets from the wallfilm due to shearing effect. The entrainment rate depends on the interfacial shear force, liquid viscosity and surface tension. Schadel-Hanratty [122] model was used to calculate the droplets generation rate. According to this model, onset of droplet generation occurs at a critical weber number. The critical Weber number, specifying the onset of entrainment is defined as:

\[ We_{cr} = \frac{\rho g U^2_{ref} \delta}{\sigma} \]

In the present study, when \( We_{cr} \geq 10.0 \), the first droplet starts to entrain into the air flow from the wall film. The model computes an entrainment mass flux which is the amount of mass sheared off the film per unit area and unit time. This rate of atomization is described by an empirical correlation, fits to data of Schadel and Hanratty which gave the model its name:

\[ R_A = X_{RA} U_{t} \sqrt{\rho g \rho_f} 10^{-3} \]
Where,

\[ X_{RA} = 0.4 \ln(150.1Riem.\text{We}_{SH} + 1) + 1.4\sqrt{I_R.\text{We}_{SH}} \]

with the roll wave intermittence factor as a function of excess film flow rate \( \Gamma_E \),

\[ I_R = 0.15 + 0.75\Gamma_E \quad \text{for} \quad I_R < 0.5 \]

In above equations an alternative definition of Weber number is used, which represents the conditions more precisely at the phase interface by replacing relative velocity with friction velocity \( U_f \):

\[ We_{cr} = \frac{\rho_g U_f^2 \delta}{\sigma} \]

The size of the generated droplets was calculated using the kataoka droplet correlation model [123]. In this model, a volume mean diameter \( D_{vm} \) of the entrained droplets fits experimental data by various authors. The maximum diameter is given as approximately 1.1 times the mean diameter which is evaluated as:

\[ D_{vm} = \frac{0.028\sigma}{\rho_g U_f^2} Re_f^{-1/3} Re_g^{2/3} \left( \frac{\rho_g}{\rho_f} \right)^{-1/3} \left( \frac{\mu_g}{\mu_f} \right)^{2/3} \]

Here, \( Re_g \) and \( Re_f \) is the gas and film Reynolds number respectively. The film Reynolds number is defined as:
Where \( \nu_f \) is the kinematic viscosity of the film liquid.

Once the droplets are torn off from the wallfilm with a certain diameter, the trajectory of these droplets was calculated using the Lagrangian approach. The interaction between the droplets were taken into account using Schmidt model \([121]\). The differential equation for the trajectory and velocity of a particle parcel in Lagrangian method is presented below:

\[
m_d \frac{dU_{id}}{dt} = F_{idr} + F_{ig} + F_{ip}
\]

Where \( F_{idr} \) is the drag force, given by:

\[
F_{idr} = \frac{1}{2} \rho_g A_d C_D |U_{rel}|
\]

Where \( A_d \) is the cross-sectional area of the particle and \( C_D \) is the drag coefficient defined by:

\[
C_D = \begin{cases} 
\frac{24}{Re_d}(1 + 0.15Re_d^{0.687}) & \text{for } Re_d < 10^3 \\
0.44 & \text{for } Re_d > 10^3 
\end{cases}
\]

Where \( Re_d \) is the particle Reynolds number and is defined as

\[
Re_d = \frac{\rho_g |U_{rel}| D_d}{\mu_g}
\]
Where \( D_d \) is the diameter of the particle.

\( F_{ig} \) is a force including the effect of gravity and buoyancy:

\[
F_{ig} = V_p (\rho_p - \rho_g) g_i
\]

Where \( V_p \) and \( \rho_p \) are the volume and density of the particle.

\( F_{ip} \) is the pressure force given by:

\[
F_{ip} = V_p \nabla P
\]

All of these models are available in the AVL FIRE 2009.2 simulation software which was used in the present study.

Figure C-2 (a) and (b) shows the predicted liquid droplets generated from jetting into the cavities at 45 and 35 degree lance inclination angles respectively. The figure shows that at 45 degree lance angle more liquid droplets splashed forward compared with 35 degree lance angle. This was expected and the reasons behind this have been well explained in section 5.4.2. The figure also shows that the size of most generated droplets was in the range of 0.1 to 0.3mm. Some droplets in the range of 0.3 to 1mm were also generated. The predicted droplets sizes are in the range of reported values of 0.04 to 5mm [118]. For better representation of the droplets diameter, the maximum range was set to one. The red particles represent droplet size of 1mm or greater. or more. The maximum predicted droplet size in the present study was determined by averaging the predicted maximum droplet diameter at every time step. This was found to be around 5mm and 4mm at 35 degree and 45 degree lance angles respectively.
Figure C-2 Distribution of generated droplets using lagrangian approach at (a) 45 degree and (b) 35 degree.
The mass of the droplets entering the splash collectors (Figure C-1) were calculated from time \( t = 5 \text{ sec} \) to 10 sec and then averaged to predict the wall splashing rate. Table C-1 shows both the calculated and measured total wall splashing rates.

Table C-1 Comparison of wall splashing rates.

<table>
<thead>
<tr>
<th>Angle</th>
<th>CFD splashing rates (g/s)</th>
<th>Experimental splashing rates (g/s)</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>35 degree</td>
<td>0.21</td>
<td>0.34</td>
<td>38</td>
</tr>
<tr>
<td>45 degree</td>
<td>1.04</td>
<td>1.09</td>
<td>5</td>
</tr>
</tbody>
</table>

At 45 degree lance angle, the calculated splashing rate is in good agreement with the experimentally measured rates with only 5% deviations. However, in case of 35 degree lance angle, the CFD model under-predicted the experimental measurement by 38%. The principle reason behind this might be the assumed cavity shape in this study. Although moving grid was used to represent the dynamic behaviour of the cavity surface, it is very difficult to completely characterize the dynamic cavity oscillations. Figure C-3 shows the splashing rate distribution on the furnace wall. Although the total splashing rate at 45 degree lance angle was well predicted by the CFD model, Figure C-3 (a) shows that, it over-predicted splashing distribution close to the water surface and underpredicted it at the upper region of the wall. But at 35 degree lance angle, the predicted wall splashing distribution was very random as shown in Figure C-3(b). The predicted maximum splashing rate was at the top region whereas in the experimental study, the splashing rate was maximum at the lower region. Again the reason is the failure in representing the complex dynamic shape of the cavity using moving grid. At 35 degree lance angle the oscillations of the cavity surface were more random compared with the 45 degree lance angle and hence, is more difficult to represent using the moving grid. The splashing rates as well as distribution can be further improved by more accurate dynamic modelling of the moving cavity shape.
Figure C-3 Wall splashing rate distribution at (a) 45 degree and (c) 35 degree lance angle.