2013

Applications of computational fluid dynamics modelling in underground coal mines

Zhongwei Wang

University of Wollongong, zw702@uowmail.edu.au

UNIVERSITY OF WOLLONGONG
COPYRIGHT WARNING

You may print or download ONE copy of this document for the purpose of your own research or study. The University does not authorise you to copy, communicate or otherwise make available electronically to any other person any copyright material contained on this site. You are reminded of the following:

This work is copyright. Apart from any use permitted under the Copyright Act 1968, no part of this work may be reproduced by any process, nor may any other exclusive right be exercised, without the permission of the author.

Copyright owners are entitled to take legal action against persons who infringe their copyright. A reproduction of material that is protected by copyright may be a copyright infringement. A court may impose penalties and award damages in relation to offences and infringements relating to copyright material. Higher penalties may apply, and higher damages may be awarded, for offences and infringements involving the conversion of material into digital or electronic form.

Unless otherwise indicated, the views expressed in this thesis are those of the author and do not necessarily represent the views of the University of Wollongong.

Recommended Citation


Research Online is the open access institutional repository for the University of Wollongong. For further information contact the UOW Library: research-pubs@uow.edu.au
Applications of Computational Fluid Dynamics Modelling in Underground Coal Mines

Zhongwei Wang

This thesis is presented in fulfilment of the requirement for the Award of the Degree of Doctor of Philosophy of the University of Wollongong

October 2013
AFFIRMATION

I, Zhongwei Wang, declare that this thesis, submitted in fulfilment of the requirements for the award of Doctor of Philosophy, in the School of Civil, Mining and Environmental Engineering, University of Wollongong, is wholly my own work unless otherwise referenced or acknowledged. The document has not been submitted for qualifications at any other academic institution.

Zhongwei Wang
10/2013
PUBLICATIONS AND AWARD

Published papers:


Completed reports:


Award:

T. Ren, Z. Wang and G. Wang. 2012 Australian Bulk Handling Review Awards for Dust Control Technology, Application or Practice - An award for best practice in dust and fume suppression, management or control; or for innovation in dust control technology, equipment or application.
ABSTRACT

Coal mine gas, spontaneous combustion and dust encountered in underground coal mines have long been recognised as safety and health hazards to mine operators. Current solutions to these issues may involve extensive pre- and post gas drainage practice, goaf inertisation strategies, dust suppression with water sprays or scrubbers. However, the effect of these control measures varies significantly from site to site owing to the lack of a fundamental understanding of the flow characteristics of the hazardous gases and dust particles under various mining conditions.

With the development of computer technology, numerical solutions to the governing equations of fluid flow and its related processes can be obtained approximately by means of the Computational Fluid Dynamics (CFD) modelling technique, which has been accepted as an indispensable predictive and design tool in almost every branch of fluid dynamics and engineering fields. A comprehensive literature review of the CFD applications in understanding the gas, spontaneous heating and dust issues for underground coal mines has been carried out, indicating the necessity of further exploring the CFD technique in solving these issues faced by underground coal mines.

Bearing in mind the principles of the CFD modelling technique, studies were first conducted at a longwall face, where both airflow and methane distribution along the face were investigated. It is notable that, for the first time, longwall CFD models were developed close to the actual longwall face geometry by incorporating the key features of the longwall equipment, for example, the belt conveyor and Beam Stage Loader (BSL) in the Maingate (MG), the chocks, Armoured Face Conveyor (AFC), spill plate and the shearer with drums. A part of the longwall goaf immediately behind the longwall chocks was also included in the longwall models to investigate the air and methane exchange between face and goaf. Furthermore, the shearer position and its cutting sequence were taken into account to evaluate their impact on face ventilation and methane flow patterns. In total, six longwall models were developed to represent different longwall operation scenarios, namely, when the
shearer was cutting close to the MG, in the middle of face and close to the Tailgate (TG) in both cutting directions.

To validate the longwall model results and assess the feasibility of investigating the longwall airflow patterns using similar CFD models, the MG-TG Case 1 (the shearer was cutting from MG towards TG and was located close to the MG) was taken as the base model and the predicted results were compared against field measured velocity profiles. The good agreement between model results and field data confirmed the reliability of base model results and further enhanced the confidence in conducting a series of parametric studies. In addition to the shearer position and its cutting direction, the impact of MG curtain and goaf caving conditions on face ventilation were also investigated. More parameters were studied to understand the methane flow characteristics on longwall face, including the flow rate, coal seam gas content, adjacent gas bearing strata, ventilation scheme and the drum sprays. Through these CFD investigations, some insights into the characteristics of airflow and gas dispersion were obtained, which would be of great significance to ventilation and gas management on longwall face. Specifically, the models’ results indicated that:

- Flow separation occurs at both the MG and TG inner corner where the flow boundary changes sharply, leading to the recirculation of airflow and subsequently the accumulation of methane gas at the two corners;
- There is significant air leakage to the goaf as airflow enters the face from the MG;
- The majority of methane trapped in the inner TG corner comes from the longwall face while the methane accumulated at the upper TG corner is caused by goaf methane emission;
- The rotation of drums affects the airflow patterns around the drums and thus helps the dilution of methane in its vicinity;
- Irrespective of the cutting sequence, the methane concentration at the TG drum is higher than it at the MG drum, and is the highest along the face;
- Methane emitted from the goaf is likely to accumulate at the upper TG corner where the airflow is approximately stagnant, and an additional TG cut
through behind face would be efficient in diluting the methane at the upper TG corner;

- The methane distribution along the face is greatly affected by the coal seam geological conditions which determine the methane sources and the corresponding emission amount, ventilation system and the flow rate at the longwall face.

Following the simulation of gas flow on longwall faces, the focus was transferred to the gas flow and migration in longwall goaf(s). Both single active longwall goaf and interconnected large longwall goafs were considered to investigate the gas distribution patterns from a perspective of spontaneous heating prediction. Due to the inaccessibility of goaf, tube bundle systems were used to monitor the gas distribution along the perimeter of goaf(s) so as to detect any gaseous products of spontaneous combustion, and these monitoring results were employed for the validation of base models. For the gas flow in a single goaf, the spatial distribution of spontaneous combustion prone zones were identified and mapped, and various control measures, which could be used to minimise the spontaneous heating zones, were evaluated with particular attention to the optimisation of proactive inert gas injection strategies. Modelling results indicated that an optimum goaf inertisation could be achieved by pumping inert gas at least 100 m behind the face on the belt road side, or ideally via surface goaf hole(s). Goaf inertisation on the retaining wall side would only be effective for localised heating, and should be used in combination with other control measures to minimise air dilution in these areas. For the multi-goaf model, both the seam elevation and the position of the active longwall face were taken into account. Areas where spontaneous heating were most likely to occur were identified at different longwall retreating stages. Then parametric studies were carried out to investigate a number of operational scenarios and their impact on goaf gas migration behaviour. Observations obtained from the model results provided insightful solutions/guidelines to the management of gas flow in longwall goafs, in particular to the early detection of spontaneous heating and its control. Specifically, the results of multi-goaf models demonstrated that:
• Overall goaf gas flow patterns change as the operating longwall retreats owing to the changing ventilation pressure differential across the entire goaf areas as well as the varying goaf elevations along the longwall panels;

• Oxygen penetration into the active goaf remains high as the longwall retreats from the start-up position to finish-off line, reaching 15% or above even some 800 m behind the longwall face;

• If roof fall or roadway failure occurs in the perimeter road and restricts ventilation, there would be more serious air ingestion into the goaf before the roof fall position;

• The most likely areas liable to the development of spontaneous combustion would be in the active goaf, the areas around the start-up seals of LW1-2 and LW4-5 and the adjacent goafs of LW7-8 at the middle part of the sealed areas;

• The gaseous products of spontaneous heating, such as CO and C₂H₆, behave similarly with most of the gaseous products dissipating into the goaf, gradually to appear along the seals of the perimeter roadway on the TG side of LW1;

• It would be difficult to detect an active goaf heating (on the MG side) at its earlier stage by solely depending upon CO/other gaseous products readings in return airflow, as the main stream of the gaseous product will be seeping out via seals along the perimeter road as well as dissipating into the sealed deep goafs in adjacent panels;

• Face ventilation has a major impact on the dispersion of gaseous products;

• Ventilation flushing of the perimeter road can help push the build up of high concentration methane into the deep goaf areas, however, this method is likely to cause excessive air leakage in certain parts of the sealed goaf;

• Goaf inertisation can be better achieved by pumping inert gas such as nitrogen at deeper positions (>200m) behind the operating longwall; the use of in-seam drainage methane for goaf inertisation will not be effective due to the buoyancy effect.

CFD simulation studies were conducted to investigate the respirable dust flow behaviour for improved dust control at the longwall entrance and ventilation intake roadways. The dispersion of respirable dust particles in these areas was reproduced
using the Lagrangian particle tracking method and the model results have been well correlated to the field monitoring results. Supported by a theoretical analysis of the respirable dust capture mechanism using ultra-fine water mist, CFD models were employed to optimise the operating conditions of a new water mist generating system in conjunction with field trials at mine sites to evaluate the performance of corresponding dust mitigation efficiency. At both the longwall entrance and the intake roadway above an underground bin, promising dust mitigation effects had been demonstrated by field dust monitoring data, indicating that over 30% and 40% of respirable dust had been reduced using the water mist system, respectively. The usefulness of CFD modelling technique in the successful design of site specific dust mitigation systems were demonstrated in these studies.

This thesis has further explored the applications of the CFD modelling technique to solve the gas, spontaneous combustion and dust issues encountered in underground coal mines. Through these specific applications, some fundamental insights into the underlying flow dynamics associated with these issues, which could not be achieved using traditional methods, were obtained from a perspective of investigations for the development of effective control measures, thus contributing significantly to the long term improvement of safety and health environment for underground coal miners.
I would like to express my deepest gratitude to my supervisors, Dr. Ting Ren and Dr. Jan Nemcik, for their invaluable guidance and critical comments as well as their consistent support throughout this study. Financial support and encouragement from Dr. Ting Ren for numbers of international and national conferences are greatly appreciated. Not only limited to my academic study, Dr. Ren’s considerate care in life also makes me enjoy my PhD study in Australia. The three years of study with Dr. Ren and Dr. Nemcik not only forms my beautiful memories but also constitutes a precious wealth in my life.

I am thankful to Professor Naj Aziz for sharing experience and making precious suggestions in the course of my study.

Special thanks go to the mine staff at Mandalong and Metropolitan collieries for providing mine site information, monitoring data and assistance in the underground ventilation survey and investigations.

I owe my sincere gratitude to Dr. Brian Plush who has assisted me significantly in completing a series of field monitoring work for the purpose of model validation and dust mitigation assessment.

I am grateful to the Australian Coal Association Research Program (ACARP) for the support of project C18019 entitled “Improved Dust Control on Longwalls Using a New Water Mist Venturi System” during the study.

I would also like to show my gratitude to Mr Graeme Cooper, Tecpro Australia, for his expert comments and insightful suggestions on the design of dust mitigation system.

I would like to express my special thanks to Bob Kininmonth for his careful and rigorous corrections on the thesis, which has improved the thesis significantly.
I would like to extend my gratitude to all my fellow students and friends, especially, Xiaotao Ma, Dr. Lei Zhang, Dalin Cai, Dr. Zhenyu Zhang, Dr. Xueyu Geng, Yu Han, Bob De Jong, Dr. Chen Cao, Gaofeng Wang, Zhenju Shan, Shuqi Ma and Qiuqiu Qiao for their friendly support and motivation during the course of this study, as well as their kind considerations in life.

I also appreciate the financial support from the University of Wollongong and the China Scholarship Council (CSC) for providing me with tuition fee and living allowance respectively.

Finally, I would like to thank my loved father, mother, and sisters for their consistent encouragement and support. They have endured so much that I could be able to continue and focus on my study. I dedicate this thesis to them.
TABLE OF CONTENTS

Affirmation ...................................................................................................................... i
Publications and award ..................................................................................................... ii
ABSTRACT ....................................................................................................................... iv
ACKNOWLEDGEMENTS ................................................................................................. ix
TABLE OF CONTENTS .................................................................................................. xi
LIST OF FIGURES ......................................................................................................... xvi
LIST OF TABLES ............................................................................................................. xxiv
1 Introduction ................................................................................................................ 1
   1.1 Background ............................................................................................................. 1
   1.2 Objectives .............................................................................................................. 5
   1.3 Scope of work ......................................................................................................... 5
   1.4 Thesis outline ........................................................................................................ 6
2 Literature review ......................................................................................................... 9
   2.1 Introduction ............................................................................................................. 9
   2.2 Longwall face methane emission prediction ....................................................... 9
      2.2.1 Traditional prediction method based on empirical correlations .......... 10
      2.2.2 Advanced prediction techniques based on numerical simulation .... 14
   2.3 CFD modelling of ventilation and gas flow in underground coal mines ....... 15
   2.4 Spontaneous combustion of coal and its control in underground coal mines
      ................................................................................................................................. 23
      2.4.1 Spontaneous combustion of coal ................................................................. 23
      2.4.2 Methods to investigate the spontaneous combustion of coal and its
          control ................................................................................................................... 24
      2.4.3 CFD investigations of spontaneous combustion and its control ...... 26
   2.5 Dust issue and its control in underground coal mines ........................................... 30
      2.5.1 Dust issue in underground coal mines ......................................................... 30
      2.5.2 Approaches to investigate the dust flow behaviour .................................. 31
      2.5.3 General practices for dust control ............................................................... 32
      2.5.4 CFD modelling of dust flow behaviour ...................................................... 34
   2.6 Summary ............................................................................................................... 37
3 Introduction to Computational Fluid Dynamics ......................................................... 38
   3.1 Introduction ............................................................................................................. 38
3.2 Governing equations
3.2.1 Continuity equation
3.2.2 Momentum equation
3.2.3 Energy equation
3.2.4 Equations of state
3.2.5 Generic transport equations
3.3 Discretisation method
3.4 Numerical method
3.5 Mesh generation
3.6 Boundary conditions
3.6.1 Inlet boundary conditions
3.6.2 Outlet boundary conditions
3.6.3 Wall boundary conditions
3.7 Post processing
3.8 Errors and uncertainties
3.9 Summary

4 Modelling of air flow patterns on a longwall face
4.1 Introduction
4.2 Development of longwall CFD models
4.3 Modelling fluid flow in underground ventilation system
4.3.1 Flow regimes in the underground ventilation system
4.3.2 Turbulence model selection
4.3.3 Transport equations for the Standard $k-\varepsilon$ model
4.4 Base model validation and results
4.4.1 Base model validation
4.4.2 Base model results
4.5 Parametric studies
4.5.1 Impact of shearer position
4.5.2 Impact of cutting sequence
4.5.3 Impact of curtain on MG side
4.5.4 Impact of goaf compaction status
4.6 Summary

5 Modelling of methane emission and flow characteristics on a longwall face
7.2 General information of the mine ................................................................. 163
  7.2.1 Mandalong mine .................................................................................. 163
  7.2.2 Longwall goaf geometry ..................................................................... 166
  7.2.3 Longwall ventilation ........................................................................... 167
  7.2.4 Gas reservoir characteristics and gas emission monitoring .............. 169

7.3 Development of CFD longwall models ................................................ 174

7.4 Base CFD model results ........................................................................ 177
  7.4.1 Base Model 1 - Operating longwall (LW9) at start-up stage .......... 177
  7.4.2 Base Model 2 - Operating longwall (LW9) at middle stage .......... 181
  7.4.3 Base Model 3 - Operating longwall (LW9) at finish-off stage ........ 183
  7.4.4 Summary of base model results ......................................................... 186

7.5 Parametric studies .................................................................................. 187
  7.5.1 Ventilation impact on the gas flow patterns in the goaf area ............ 187
  7.5.2 Impact of roof fall in the perimeter road ........................................... 190
  7.5.3 Spontaneous combustion gaseous products flow patterns ............. 194
  7.5.4 Impact of face ventilation on CO and C2H6 flow pattern ................. 196
  7.5.5 With regulator applied in new LW TG Active Goaf upon Sealing .... 199
  7.5.6 Sealing flank and effect upon sealed goaf ......................................... 202
  7.5.7 Flood ventilation of flank/perimeter road ........................................ 203
  7.5.8 Goaf inertisation using nitrogen (N2) .............................................. 206
  7.5.9 Goaf inertisation using in-seam drainage methane ......................... 207

7.6 Summary .................................................................................................. 209

8 Improved respirable dust control for dust from the mg chocks and bsl .... 212
  8.1 Introduction ............................................................................................ 212
  8.2 The water mist based venturi unit(s) for respirable dust mitigation at
      longwall faces .......................................................................................... 214
    8.2.1 Respirable dust capture mechanism using the water mist technology 214
    8.2.2 Prototype of the newly developed water mist based venturi unit(s) for
         longwall chocks .................................................................................... 216
  8.3 Equations governing the respirable dust flow ....................................... 217
  8.4 CFD modelling of respirable dust flow patterns .................................... 219
  8.5 Impact of water mist on respirable dust flow patterns and optimisation of
      sprays operating condition .................................................................... 224
LIST OF FIGURES

Figure 1.1 World coal production by year (Anon., 2013) ....................................................... 1
Figure 2.1 Gas emission methods used in Germany (Noack, 1998) ................................. 10
Figure 2.2 Various methods of determining strata degassing coefficients (Lunarzewski, 1998) ................................................................................................................. 11
Figure 2.3 Methane concentration contour and airflow pathlines in a heading face (Wala, et al., 2008) ........................................................................................................... 17
Figure 2.4 Methane flow with one drainage borehole (Kelsey, et al., 2003) .............. 18
Figure 2.5 Velocity vector distribution in a heading (Hargreaves and Lowndes, 2007) ................................................................................................................................. 19
Figure 2.6 Methane concentration distribution on a cross section (Toraño, et al. 2009) ............................................................................................................................... 20
Figure 2.7 Geometry of longwall face model (Zheng and Tien, 2009) .................. 20
Figure 2.8 CFD modelling of roof boreholes intercepting seam gas from overlying seams (Ren and Balusu, 2010) .......................................................... 21
Figure 2.9 Determination of explosive gas mixture range (Worra ll, 2012) ........... 22
Figure 2.10 Velocity contours in the cross-cut region (Sasmito, et al., 2013) ....... 23
Figure 2.11 Impact of inert gas injection on oxygen distribution in the goaf (Balusu, et al., 2002) ................................................................................................................. 27
Figure 2.12 Temperature field in one modelled scenario (Lolon, 2008) ............... 28
Figure 2.13 Temperature distribution in the goaf area with face advance rate at ...... 28
Figure 2.14 Oxygen contour in longwall goafs (Yuan and Smith, 2008b) ............. 29
Figure 2.15 Relationship between air velocity and dust concentration (Vutukuri and Lama, 1986) .............................................................................................. 33
Figure 2.16 Respirable dust flow patterns near the shearer (Balusu, et al., 2005a) .... 35
Figure 2.17 Impact of shearer scrubber on dust flow behaviour (Ren and Balusu, 2010) ......................................................................................................................... 35
Figure 2.18 Dust concentration isocontours in mg/m$^3$ on the cross section located.. 36
Figure 3.1 One-dimensional and two-dimensional uniformly distributed Cartesian grids for the FDM (full symbols denote boundary nodes and open symbols denote computational nodes) (Tu, et al., 2007) .................................................... 43
Figure 3.2 A representation of structured and unstructured mesh for the FVM (full symbols denote element vertices and open symbols at the centre of the control volumes denote computational nodes) (Tu, et al., 2007) ................................. 44
Figure 3.3 The SIMPLE algorithm solution procedure ........................................ 45
Figure 3.4 A typical convergence history during a calculation .............................. 46
Figure 4.1 Layout of the longwall model with shearer cutting from MG to TG ...... 58
Figure 4.2 Longwall model with shearer cutting from TG to MG....................... 58
Figure 4.3 Computational grids adopted for the TG-MG Case 1 longwall model .... 59
Figure 4.4 Validation of CFD model – predicted airflow velocity vs. field ventilation survey data ........................................................................................................ 68
Figure 4.5 The laminar boundary layer generated on the surface of an object....... 70
Figure 4.6 The flow separation process (Herbert, et al., 2009)............................. 71
Figure 4.7 Flow schematic at the MG corner..................................................... 73
Figure 4.8 Flow patterns at the MG corner indicating airflow separation - 2m above floor level .................................................................................................................. 73
Figure 4.9 Flow patterns at the TG corner indicating airflow separation - 2m above floor level .................................................................................................................. 74
Figure 4.10 Velocity contour at different levels above floor ............................... 76
Figure 4.11 Airflow streamlines (form both the MG and the MG ct) .................... 78
Figure 4.12 Airflow streamlines from the MG belt roadway only ...................... 79
Figure 4.13 Airflow velocity contour around the shearer ................................. 81
Figure 4.14 3D view of airflow velocity vectors around the shearer ................. 82
Figure 4.15 The impact of drum rotation on airflow patterns (blue arrow indicates drum rotation direction) ................................................................. 83
Figure 4.16 Velocity distribution at different locations across the face - 2 m above the floor ........................................................................................................... 84
Figure 4.17 Velocity contour at 2 m above floor for MG-TG Case 2 .............. 85
Figure 4.18 Velocity contour at 2 m above floor for MG-TG Case 3 .............. 85
Figure 4.19 3D view of velocity vectors around the shearer for MG-TG Case 2 .... 86
Figure 4.20 3D view of velocity vectors around the shearer for MG-TG Case 3 .... 87
Figure 4.21 Velocity contour at different levels above floor for TG-MG Case 1 .... 89
Figure 4.22 3D view of velocity vectors around the shearer for TG-MG Case 1 .... 90
Figure 4.23 Velocity contour on the surface 2 m above floor for TG-MG Case 2 .... 91
Figure 5.14 Methane concentration contour at 2 m above floor with 3.5 m$^3$/s flow rate through the TG cut through................................................................. 127
Figure 5.15 Spatial distribution of 1% methane concentration iso-surface with 17.5 m$^3$/s flow rate through the TG cut through ................................................................. 128
Figure 5.16 Spatial distribution of 1% methane concentration iso-surface with 10.5 m$^3$/s flow rate through the TG cut through ................................................................. 129
Figure 5.17 Spatial distribution of 1% methane concentration iso-surface with 3.5 m$^3$/s flow rate through the TG cut through ................................................................. 129
Figure 5.18 Methane concentration distribution around the shearer (left: base model; right: with drum sprays on) ........................................................................................................ 131
Figure 5.19 Methane concentration distribution around the TG drum (left: base model; right: with drum sprays on) ........................................................................................................ 132
Figure 5.20 Methane concentration distribution around the MG drum (left: base model; right: with drum sprays on) ........................................................................................................ 132
Figure 5.21 Methane concentration distribution at 1.5 m above floor .................. 133
Figure 5.22 Methane concentration distribution at 2 m above floor .................. 134
Figure 5.23 Methane concentration distribution at 2.5 m above floor .................. 134
Figure 5.24 Methane concentration distribution at 2 m above floor with curtain on the MG corner ........................................................................................................ 136
Figure 6.1 Longwall face layout and ventilation system ........................................ 141
Figure 6.2 Layout of goaf gas monitoring points .................................................. 142
Figure 6.3 Setup of a typical monitoring point ..................................................... 143
Figure 6.4 Goaf gas distribution when longwall face retreated 210 m .............. 144
Figure 6.5 Goaf gas distribution when longwall face retreated 506 m .............. 144
Figure 6.6 Goaf gas distribution when longwall face retreated 848 m .............. 145
Figure 6.7 Longwall CFD model geometry and computational grid .................. 147
Figure 6.8 Base CFD model validation - model result vs. field monitoring data ... 148
Figure 6.9 Oxygen distribution and spatial distribution of spontaneous heating zones in the goaf ........................................................................................................ 150
Figure 6.10 Oxygen concentration distribution on roof level at different distances behind face (vertical to the face retreating direction) ........................................ 150
Figure 6.11 Oxygen concentration distribution on different roof level in the middle of face (from face to the start up line) .............................................................................. 151
Figure 6.12 Goaf inertisation effect on oxygen distribution - injection on belt road side .......................................................... 153
Figure 6.13 Goaf inertisation on oxygen distribution - injection from retaining wall .......................................................... 154
Figure 6.14 Goaf inertisation effect on oxygen distribution - injection behind chocks .......................................................... 156
Figure 6.15 Goaf inertisation effect on oxygen distribution - injection from surface wells/boreholes .......................................................... 158
Figure 6.16 Goaf gas distribution after ventilation adjustment .......................................................... 159
Figure 7.1 Longwall layout and topography .......................................................... 163
Figure 7.2 Typical stratigraphic and cross-section of Mandalong Mine .......................................................... 164
Figure 7.3 Longwall panel layout at Mandalong Mine .......................................................... 165
Figure 7.4 Longwall goaf geometry .......................................................... 166
Figure 7.5 Seam floor contours - West Wallarah seam .......................................................... 167
Figure 7.6 A typical ventilation system in Mandalong Mine .......................................................... 168
Figure 7.7 West Wallarah seam gas content (m$^3$/t) .......................................................... 169
Figure 7.8 Generic longwall excavation layout in relation to Fassifern seam .......................................................... 170
Figure 7.9 A snapshot of goaf gas distributions in longwall goafs - from start-up to near finish-off stage .......................................................... 174
Figure 7.10 CFD model geometry for LW9 retreating at different stages - plane view .......................................................... 175
Figure 7.11 CFD model geometry - 3D view .......................................................... 176
Figure 7.12 CFD model computational mesh .......................................................... 177
Figure 7.13 Ventilation scheme at the start-up stage .......................................................... 178
Figure 7.14 Methane and oxygen distribution pattern in longwall goafs when LW9 at start-up stage (projection from roof) .......................................................... 179
Figure 7.15 Oxygen distribution pattern in longwall goafs when the longwall (LW9 is at the start-up stage) .......................................................... 180
Figure 7.16 Ventilation scheme at the middle stage .......................................................... 181
Figure 7.17 Longwall goaf gas (methane and oxygen) distribution pattern when the longwall (LW9) has retreated to the middle part of the panel .......................................................... 183
Figure 7.18 Ventilation scheme at the finish-off stage .......................................................... 184
Figure 7.19 Goaf gas (methane and oxygen) distribution pattern when the longwall (LW9) has retreated close to finish-off line (roof projection) ........................................ 185
Figure 7.20 Longwall goaf gas (methane and oxygen) distribution pattern when LW9 has retreated close to finish-off line - 3D view .................................................. 186
Figure 7.21 Oxygen distribution pattern in goaf with increased face ventilation - start-up stage .......................................................... 188
Figure 7.22 Oxygen distribution pattern in goaf with increased face ventilation - midstage ........................................................................ 189
Figure 7.23 Oxygen distribution pattern in goaf with increased face ventilation - finish-off stage .................................................................... 190
Figure 7.24 Roof fall point in the flank road .............................................................. 191
Figure 7.25 Gas distribution pattern in goaf with roof fall in the perimeter road - model 2 ................................................................................. 192
Figure 7.26 Gas distribution pattern in goaf with roof fall in the perimeter road - model 3 ................................................................................. 193
Figure 7.27 CO flow patterns in the goaf area .......................................................... 195
Figure 7.28 CO and C₂H₆ flow patterns in the goaf area ........................................ 195
Figure 7.29 CO and C₂H₆ gas flow patterns in the goaf area - heating in active goaf ......................................................................................... 196
Figure 7.30 CO dispersion pattern with increased face ventilation ....................... 197
Figure 7.31 CO dispersion pattern with reduced face ventilation .......................... 198
Figure 7.32 Impact of ventilation on CO dispersion from a heating on the edge of sealed goaf ......................................................................................... 199
Figure 7.33 Ventilation scheme with regulator applied in the new LW TG upon sealing ........................................................................................................... 200
Figure 7.34 Goaf gas flow pattern with regulations applied in new LW TG upon sealing ........................................................................................................... 201
Figure 7.35 Ventilation system while sealing the flank ............................................ 202
Figure 7.36 Effect of sealing flank on goaf gas flow distribution pattern ............. 203
Figure 7.37 Flood ventilation of flank/perimeter road ........................................... 204
Figure 7.38 Goaf gas distribution following flooding ventilation of flank/perimeter road ......................................................................................... 205
Figure 7.39 Effect of nitrogen injection at various locations behind the face ....... 207
Figure 7.40 Effect of methane injection into active goaf .............................................. 209
Figure 8.1 Dust survey on a longwall showing MG Chocks (1-5) advance dust versus
total longwall dust make at shearer MG and TG operator positions (Gillies and
Wu, 2008, 2010) .............................................................................................................. 213
Figure 8.2 The impact of droplet size on the capture of respirable dust particles (after
Schowengerdt and Brown, 1976) .................................................................................... 216
Figure 8.3 The completed prototype water mist venturi units ........................................ 217
Figure 8.4 Layout of the longwall model .......................................................................... 220
Figure 8.5 Flow patterns of dust particles released from MG, BSL and chocks-plan
view ................................................................................................................................. 222
Figure 8.6 Flow patterns of dust particles released from MG, BSL and chocks-3D
view ................................................................................................................................. 224
Figure 8.7 The MG dust mitigation effect ............................................................................ 226
Figure 8.8 Sprays used for BSL dust mitigation ................................................................. 227
Figure 8.9 BSL dust mitigation effect .................................................................................. 228
Figure 8.10 Sprays used for MG chock dust mitigation ......................................................... 229
Figure 8.11 MG chocks dust mitigation effect ..................................................................... 231
Figure 8.12 Chock dust mitigation (immediately behind shearer) ........................................ 233
Figure 8.13 A venturi unit installed on the BSL in the MG .................................................. 234
Figure 8.14 Venturi units installed on MG Chock#6 ............................................................. 236
Figure 8.15 Venturi units installed on longwall Chock#6 in operation .................................. 237
Figure 8.16 Location of dust monitor placement along the longwall face ............................ 238
Figure 8.17 Respirable benchmark dust production ............................................................ 239
Figure 8.18 Respirable dust production benchmark versus BSL venturi on ....................... 239
Figure 8.19 BSL venturi spray respirable dust mitigation efficiency ............................... 240
Figure 8.20 Respirable dust production benchmark versus chock venturi’s operating
........................................................................................................................................... 240
Figure 8.21 The respirable dust mitigation efficiency by chock venturi units.................... 241
Figure 8.22 Total respirable dust mitigation efficiency by the installation of venturi
units at BSL and chock #6 .............................................................................................. 241
Figure 8.23 The water mist cloud travelling along the face ................................................ 243
Figure 9.1 Mine ventilation system (arrows showing the airflow direction) ....................... 246
Figure 9.2 Mine ventilation system around the underground bin ..................................... 246

xxii
Figure 9.3 Actual configurations of the underground bin ............................................. 247
Figure 9.4 Overview of the CFD model ......................................................................... 250
Figure 9.5 Comparison of predicted and measured airflow velocity profiles along the belt roadway ................................................................................................................. 252
Figure 9.6 Airflow circulation area observed during field ventilation survey ............ 252
Figure 9.7 CFD prediction of airflow pattern showing circulation area just outbye the bin ................................................................................................................................. 253
Figure 9.8 Airflow velocity distributions around the underground bin .................... 256
Figure 9.9 Air streamlines indicating the flow of air stream from the horizontal intake .......................................................................................................................... 256
Figure 9.10 Streamlines showing the generation of airflow circulation .................... 257
Figure 9.11 Airflow velocity profiles at fixed elevations across the computational domain ......................................................................................................................... 257
Figure 9.12 Dispersion of respirable dust particles ...................................................... 259
Figure 9.13 A comparison of measured and predicted dust concentrations along the belt roadway ..................................................................................................................... 260
Figure 9.14 Predicted dust concentration contour at 1.7 m above the floor ............ 261
Figure 9.15 Evolution of dust concentration along the belt roadway (4.3 m³/s from the horizontal intake) ......................................................................................... 262
Figure 9.16 Distribution of projected lines for dust concentration analysis .............. 262
Figure 9.17 Ventilation impact on dust concentration evolution along the belt roadway ......................................................................................................................... 265
Figure 9.18 Proposed sprays location and potential operating conditions ............... 266
Figure 9.19 Generation of water mist under different operating conditions .......... 267
Figure 9.20 Design of the new dust mitigation system and its projected spray coverage ............................................................................................................................. 269
Figure 9.21 Configurations of the flat fan nozzle ....................................................... 269
Figure 9.22 Impact of the new system on the respirable dust concentration along the roadway ................................................................................................................. 270
Figure 9.23 Respirable dust reduction efficiency along the roadway ....................... 270
LIST OF TABLES

Table 1.1 Top ten coal producers (Anon., 2012) .................................................. 1
Table 1.2 Major coal mine explosion incidents after 2000 (after Karacan, et al., 2011) ................................................................. 3
Table 4.1 Short name for the six longwall models ............................................. 59
Table 5.1 Determination of gas emission rate from exposed face rib ............... 104
Table 5.2 Gas emission rate around the drums and on the AFC ..................... 105
Table 6.1 The roof and floor conditions of 154307 face ................................ 140
Table 6.2 The actual ventilation rate to the 154307 face ................................. 141
Table 6.3 Location of monitoring points relative to longwall face start-up ....... 143
1 INTRODUCTION

1.1 Background

In 2011, the total production of coal in the world had reached 7678 Mt, and it was reported by World Coal Association (WCA) that coal had been used to generate 42% of the world’s electricity and provide 30.3% of global primary energy needs - the highest since 1969 (Anon., 2012). There is no doubt that the role of coal in the world development and civilization process is becoming increasingly important. Figure 1.1 shows the world coal production in recent years (Anon., 2013). The top ten coal producers in the world are shown in Table 1.1, where it can be seen that China accounts for 45.2% of world coal production. And Australia is the 4th largest coal producer in the world.

![World coal production by year](image)

**Figure 1.1 World coal production by year (Anon., 2013)**

<table>
<thead>
<tr>
<th>PR China</th>
<th>3471Mt</th>
<th>Russia</th>
<th>334Mt</th>
</tr>
</thead>
<tbody>
<tr>
<td>USA</td>
<td>1004Mt</td>
<td>South Africa</td>
<td>253Mt</td>
</tr>
<tr>
<td>India</td>
<td>585Mt</td>
<td>Germany</td>
<td>189Mt</td>
</tr>
<tr>
<td>Australia</td>
<td>414Mt</td>
<td>Poland</td>
<td>139Mt</td>
</tr>
<tr>
<td>Indonesia</td>
<td>376Mt</td>
<td>Kazakhstan</td>
<td>117Mt</td>
</tr>
</tbody>
</table>
Coal is a fossil fuel, it forms when dead plant matter is converted into peat, which in turn is converted into lignite, then sub-bituminous coal, after that bituminous coal, and lastly anthracite (Anon., 2013). This involves biological and geological processes that take millions of years.

Depending on the geological conditions, coal resources can be extracted through two main methods – surface mining and underground mining. Although surface mining is prevalent in several important coal producing countries (e.g. 80% of coal production in Australia is obtained from surface mining), 60% of world coal production comes from underground coal mines (Anon., 2009), and this can be attributed to the fact that more than 95% of the total coal production of China is provided by underground coal mines.

Both surface and underground mining have their own specific concerns, while the safety and health issues encountered in underground mines are more severe due to their unique working conditions. The majority of coal mine disasters known to the public occurred in underground working locations, i.e., the longwall faces, goafs and development headings. Among various safety issues faced by underground coal mines, the methane gas explosion is well known to the public as it always results in a large number of fatalities. Table 1.2 illustrates some major coal mine methane gas explosions that have occurred in recent years. Therefore, the alarming numbers demonstrate that more efforts are necessary to be made for better methane management in underground coal mines.

Besides the threat of methane in the underground working environment, another gas which may be encountered in the underground ventilation system is carbon monoxide, a product of spontaneous heating of coal. Carbon monoxide is a chemical asphyxiant, it affects the oxygen level in human blood significantly, causing death if breathed in excessive amounts. The same as for methane, there is a rigid requirement for the occurrence of carbon monoxide in the ventilation system. In Australia, the statutory concentration is 25 ppm (parts per million) whilst in China it is 24 ppm (Bell, 2013; Anon., 2004). Due to the limitations of underground conditions, the prediction and detection of spontaneous heating in the goaf is still difficult.
Engineers need to rely on the tube bundle system to determine the occurrence of heating and its intensity. It is acknowledged that the occurrence of spontaneous combustion in the goaf may develop into fire disasters in some cases, and subsequently the explosion of methane or coal dust under certain circumstances. A recent investigation of the fire and explosion that occurred at Blakefield South mine, Australia, again reveals that spontaneous combustion may have contributed sufficient heat to ignite the methane (Flowers and Stewart, 2012).

Table 1.2 Major coal mine explosion incidents after 2000 (after Karacan, et al., 2011)

<table>
<thead>
<tr>
<th>Country</th>
<th>Date</th>
<th>Coal Mine</th>
<th>Fatalities</th>
</tr>
</thead>
<tbody>
<tr>
<td>China</td>
<td>14/02/2005</td>
<td>Sunjiawan, Haizhou shaft, Fuxin</td>
<td>214</td>
</tr>
<tr>
<td>USA</td>
<td>2/06/2006</td>
<td>Sago, West Virginia</td>
<td>12</td>
</tr>
<tr>
<td>Kazakhstan</td>
<td>20/09/2006</td>
<td>Lenina, Karaganda</td>
<td>43</td>
</tr>
<tr>
<td>Russia</td>
<td>19/03/2007</td>
<td>Ulyanovskaya, Kemerovo</td>
<td>108</td>
</tr>
<tr>
<td>Ukraine</td>
<td>19/11/2007</td>
<td>Zasyadko, Donetsk</td>
<td>80</td>
</tr>
<tr>
<td>USA</td>
<td>5/04/2010</td>
<td>Upper Big Branch, West Virginia,</td>
<td>29</td>
</tr>
<tr>
<td>Turkey</td>
<td>17/05/2010</td>
<td>Karadon, Zonguldak</td>
<td>30</td>
</tr>
<tr>
<td>New Zealand</td>
<td>19/11/2010</td>
<td>Pike River Mine</td>
<td>29</td>
</tr>
</tbody>
</table>

Bearing in mind that methane explosions are just one of the mining induced issues which result in instant fatalities, the Coal Workers' Pneumoconiosis (CWP), commonly referred to as black lung disease, kills miners in the long term. This kind of disease is irreversible and can be debilitating, progressive, and potentially fatal in its most extreme cases. Like cancer, there is till now no cure to CWP. The increase in coal production leads to a dramatic increase in dust exposure levels in the underground working environment and without adequate protection strategies, CWP is getting prevalent among miners. In the US, it is estimated that around 1500 former miners will die from CWP every year (James, 2011). It is also reported that close to nine percent of miners with 25 years or more experience tested positive for black lung in 2005-2006, compared with four percent in the late 1990s (Anon., 2011). As a result, there is an urgent need for the development of effective dust control strategies.

It is worth noting that methane explosions, spontaneous heating and dust contamination are all highly related to the underground ventilation, and the main functions of the ventilation system are to tackle these issues. In the normal ventilation design, these issues must be taken into account, and some of them may
become the predominant factors in terms of determining the panel layout and the flow rate.

Current solutions to these issues involve extensive pre and post methane drainage practice, goaf inertisation strategies, and water sprays or dust scrubbers. However, the effect of these control measures varies significantly from mine to mine. The variation of local geological conditions accounts for an important reason. To a great extent, it can be attributed to the lack of sufficient understanding of the flow characteristics of the hazardous gas and dust under various mining conditions. For instance, on a longwall face, real time gas monitors are installed on the shearer body, at upper tailgate (TG) corner and in the return airflow of the TG; however, these monitors may not be able to detect the highest methane concentration in the vicinity of cutting drums, where frictional ignitions may occur if the methane concentration reaches the explosive range. Similarly, the incorrect operation of sprays for the purpose of dust suppression may lead to increased dust level at the longwall crews’ breathing zone.

Nowadays, with the development of computer technology, the Navier-Stokes equations governing the fluid flow can be solved numerically, based on which the Computational Fluid Dynamics (CFD) simulation has earned its popularity in almost every branch of fluid dynamics and engineering fields. It has been widely used as a powerful predictive and design tool across all industrial sectors and the mining industry is no exception. To better understand the fugitive gas and dust flow behaviour in a complex underground mining environment and to evaluate the effectiveness of various control strategies, the CFD modelling approach has become a necessity to supplement laboratory experiments and field studies. This thesis concentrates on the applications of the CFD modelling technique in solving gas and dust issues encountered in underground coal mines, demonstrating the role of this technique can play in the design and optimisation of effective gas and dust control strategies.
1.2 Objectives

The main objective of this thesis is to further explore the applications of CFD modelling technique in underground coal mines for providing detailed and predictive information on the gas and dust flow characteristics in the underground ventilation system, as well as the development of corresponding control strategies. Specifically, it involves the following aspects:

- To provide a detailed airflow field on a typical underground longwall face considering various longwall operation scenarios;
- To investigate the methane gas flow and accumulation characteristics along the longwall face taking into account the variation of mining conditions;
- To understand the gas flow patterns and distribution in both a single active goaf and a multi-goaf zone from the perspective of spontaneous combustion control;
- To investigate the respirable dust flow patterns at the longwall face due to the MG chocks advancement and intake contaminations;
- To understand the respirable dust flow patterns above an underground bin and propose an effective dust mitigation strategy on the basis of CFD modelling results.

1.3 Scope of work

To achieve the objectives of the thesis, a series of numerical modelling investigations have been carried out using the CFD technique. Field studies have also been conducted during the course of this study to collect necessary data on airflow distribution at a longwall face and goaf gas distribution for the corresponding CFD models’ validation and calibration purposes. Therefore, the scope of work includes the following aspects:

- Literature review of gas and dust related issues encountered in underground coal mines and their corresponding control strategies generally conducted in field practices;
• Development of innovative three dimensional (3D) CFD longwall models based upon realistic longwall layouts and equipment profiles such as hydraulic supports and longwall shearer by taking into account the shearer position and the cutting sequence;
• Validation of the base model predicted airflow using field data, then a detailed analysis of the base model results is conducted after which extensive parametric studies are carried out;
• Model investigations of the methane flow patterns along the face considering the shearer position and the cutting sequence, as well as the methane flow characteristics under various geological mining conditions;
• Development of a single goaf model, validation of the base model using the monitored gas composition along the goaf perimeter and parametric studies for the optimisation of effective goaf inertisation strategies;
• Development of multi-goaf CFD models considering the advance of active goaf, validation of base models and extensive parametric studies to investigate the impact of various operation scenarios on the goaf gas distribution patterns;
• Investigation of the respirable dust flow patterns following the advance of the MG chocks and BSL, optimisation of the operating conditions of sprays for better dust suppression and conduct field evaluation;
• Studies of the respirable dust flow patterns above an underground bin, development of effective respirable dust mitigation strategies and field implementation of the final dust control strategy and its assessment;
• Conclusions and recommendations for future work.

1.4 Thesis outline

The thesis is composed of ten chapters.

Chapter 1 is the overall introduction, where a brief description of the background, objectives, work scope and outline of the thesis is presented.
Chapter 2 consists of a critical literature review of the safety and health issues encountered in the underground coal mines, with special focus on the hazardous methane gas and dust. The methane emission prediction methods developed in the main coal production countries have been reviewed, as well as the major methane control strategies, ranging from ventilation, pre drainage to post drainage. The state-of-art on respirable dust control in the mining industry constitutes another significant portion of Chapter 2.

Chapter 3 summarises fundamentals of the CFD technique, where the governing equations of fluid flow is introduced, and a typical solution process using the CFD technique is described, together with a review of the current status of CFD applications in the mining industry.

Chapter 4 describes the use of CFD models to predict the airflow patterns on a typical longwall face. Upon the validation of the base model, a detailed airflow field along the face has been obtained, especially at the MG, TG corners and in the vicinity of the shearer where a detailed analysis is provided. Then, parametric studies are conducted to investigate the impact of various operation scenarios on the airflow patterns along the face.

Chapter 5 investigates the methane flow behaviour along the face on the basis of flow field obtained from Chapter 4. The methane emission rate at different positions along the face and in the immediate goaf is determined through the mathematical model proposed by Airey (1971); meanwhile, the decay of methane emission from the exposed ribs is also taken into account. A detailed analysis of methane flow characteristics along the face has been presented as the variation of shearer position and cutting sequence. The influence of geological conditions has been investigated through a set of parametric studies.

Chapter 6 presents the use of CFD models to predict the gas distribution within an active goaf and the optimisation of effective goaf inertisation strategies. Field monitored data have been used to validate the base model results, after which a spatial distribution of the spontaneous combustion prone zone in the goaf is predicted.
and visualised. An assessment has been conducted among various inert gas injection strategies to determine the optimum goaf inertisation strategy.

Chapter 7 describes the gas distribution patterns in a multi-goaf and the impact of various operational conditions on its migration patterns in the goafs. Three base models have been constructed considering the advancement of the active goaf. General flow patterns of the goaf gases have been obtained. Extensive parametric studies have also been conducted to investigate the behaviour of goaf gases under various operational conditions.

Chapter 8 investigates the new respirable dust mitigation strategy that can be used for dust generated from the MG chocks movement, BSL and intake activities. The CFD models are used to optimise the operating conditions of the water mist based sprays in terms of operating direction. Guided by the model results, field tests have been carried out in an underground coal mine where a promising respirable dust mitigation effect has been demonstrated.

Chapter 9 presents the use of CFD models to investigate the air and respirable dust flow characteristics above an underground bin, through which an innovative respirable dust mitigation strategy has been proposed and implemented on site. Further field evaluations have indicated the excellent performance of the new respirable dust mitigation system.

Chapter 10 summaries the main conclusions achieved in the thesis. Recommendations for future work are also included in this chapter.
2 LITERATURE REVIEW

2.1 Introduction

Gas emission and dust liberation are unavoidable issues encountered in the extraction of coal resources, and these issues are of great importance in underground coal mines owing to the limitation of ventilation systems and the absence of adequate control measures in the confined working space. If not managed properly, they will bring the potential safety and health threat to the workforce into reality via various forms. Therefore, it is essential to understand the occurrence of methane, spontaneous heating and dust issues in underground coal mines before conducting the corresponding CFD investigations for improved understanding and better solutions. As a result, this chapter first reviews the main methods developed worldwide for longwall face methane emission prediction. Then, investigations of the ventilation and gas flow modelling in underground coal mines are reviewed. Further, the review of spontaneous combustion of coal and its control is carried out, and the CFD application for the spontaneous combustion control is also included. Finally, the dust issue and its control, including the generally developed dust control methods and the CFD applications, are reviewed.

2.2 Longwall face methane emission prediction

As the main production section, the longwall face is undoubtedly the largest contributor to methane emission of a mine and more focus has been put on it. During longwall extraction, large quantities of released methane come from not only mined seam, but also from adjacent gas bearing strata within a certain distance from mined seam. Such gas migrates into the working environment, and research indicates that as much as 90% and sometimes even up to 97% of methane entering a longwall panel may come from adjacent seams, especially the overlying seams (Whittles, et al., 2006; You, et al., 2008). To determine the quantity of methane emitted from adjacent seams, a variety of methods have been developed.
2.2.1 Traditional prediction method based on empirical correlations

Based on the degree of gas emission and residual gas pressure, the PFG/FGK (PFG represents the degree of gas emission curve for the roof whilst FGK stands for the degree of gas emission curve for the floor) method and the gas pressure method have been employed in German coal mines for methane emission from the roof and floor (Noack, 1998). Both methods take into account not only the adsorbed gas but also the free gas, and the Langmuir’s sorption isotherm is used to calculate residual gas content in the gas pressure method. Figure 2.1 shows the determination of gas emission degree and residual gas pressure in the roof and floor. As can be seen from the PFG/FGK method, in the absence of gas emission measurements, a mean degree of gas emission of 75% of the gas in the mined coal seam is assumed, in an area of 20 m above and 11 m below the mined coal seam, the gas emission degree is assumed to be 100%.

Figure 2.1 Gas emission methods used in Germany (Noack, 1998)

Figure 2.2 illustrates various methods developed for the determination of the degassing coefficient of roof and floor. It can be seen that researchers have different views on the degree of gas emission from overlying and underlying strata, and this reflects different physical models adopted under different mining methods and site conditions.
specific geological conditions (Lunarzewski, 1998; Yu, 1992). However, all of them accept that some factors strongly affect the gas emission prediction, including mining methods, rock properties above and below the mined coal seam, gas content in the gas bearing strata and seams, geological structures such as faults and dykes, and the dip angle of the strata.

Figure 2.2 Various methods of determining strata degassing coefficients (Lunarzewski, 1998)

Lunarzewski and Battino (1983) proposed that the quantity of methane released from both the working seam and adjacent gas bearing strata can be determined by the following formula:

\[ Q_T = W_M + Q_1 + Q_2 \]  \hspace{1cm} (2.1)

where, \( Q_T \) is the total methane emission, m\(^3\)/t; \( W_M \) is the methane quantity released from the mined coal seam, m\(^3\)/t; \( Q_1, Q_2 \) are methane emission from overlying and underlying seams, respectively, m\(^3\)/t. And \( Q_1, Q_2 \) can be calculated from the following equation:

\[ Q_{1,2} = \sum \frac{m_A \times W_A \times \theta_A}{100m_M} \]  \hspace{1cm} (2.2)
where, \( m_A \) is the thickness of investigated adjacent seams, m; \( W_A \) is methane content of investigated adjacent seams, \( m^3/t \); \( \theta_A \) is degassing coefficient of investigated adjacent seams, \( \% \); \( m_M \) is the thickness of mined seam, m.

Yu et al. (2000) indicates that methane released from rib, excavated coal and goaf are the main contributors to longwall face emission. For the first two parts, the following equation is proposed:

\[
q_m = C \times X \quad (2.3)
\]

where, \( q_m \) is the relative methane emission from mined coal seam, \( m^3/t \); \( X \) is the methane content of mined coal seam, \( m^3/t \); \( C \) is the degassing coefficient of mined coal seam, and is generally between 0.4 and 0.8 for coal seams with thin and medium thickness.

Methane emitted from goaf is dependent on the ventilation system adopted, the methane content of adjacent strata, the advance rate of face, the thickness of gas bearing strata and the corresponding emission degree similar to the PFG/FGK method, thus, it can be stated as:

\[
q_g = C_g \times \alpha v^c \frac{mL\delta}{24 \times 60} \sum_{i=1}^{n} \frac{m_i x_i \eta_i}{m} \quad (2.4)
\]

where, \( C_g \) is the coefficient determined by ventilation system (\( \leq 1 \)); \( v \) is the advance rate of face, m/d; \( \alpha v^c \) is the correction factor taking into account the lag of roof and floor break and displacement relative to time and space; \( m_i, x_i \text{ and } \eta_i \) are the thickness, gas content and emission degree of adjacent gas bearing strata, respectively.

Taking into account the residual methane content, another formula has been put forward (Yu, 2005; Cheng, et al., 2006):
\[ q = q_m + q_a = k \frac{M}{m} (X_0 - X_c) + \sum_{i=1}^{n} \frac{M_i}{m} \eta_i (X_{0i} - X_{ci}) \]  

(2.5)

\[ k = k_1 \cdot k_2 \cdot k_3 \]  

(2.5a)

\[ k_2 = \frac{1}{c} \]  

(2.5b)

\[ k_3 = \frac{(L - 2b)}{L} \]  

(2.5c)

where, \( q, q_m \) and \( q_a \) are methane emission of longwall face, mined seam and adjacent strata, respectively; \( k \) is the influencing coefficient of methane emission from mined seam; \( k_1 \) is coefficient of methane emission from surrounding strata, and \( k_1 = 1.2 \) for fully caving; \( k_2 \) is coefficient of methane emission from coal left in face; \( c \) is the recovery ratio of face, \%; \( k_3 \) is influencing coefficient of pre-methane emission in the process of roadway development, \( L \) is face length, \( m \); \( b \) is width of pre-methane emission in roadway, \( m \); \( m \) and \( M \) are mining height and seam height respectively, \( m \); \( X_0 \) is the initial methane content of coal seam, \( m^3/t \); \( X_c \) is residual methane content of coal seam, \( m^3/t \); \( M_i \) is thickness of the \( i \) adjacent strata, \( X_{0i} \) is the initial methane content of \( i \) adjacent strata, \( m^3/t \); \( X_{ci} \) is residual methane content of the \( i \) adjacent strata, \( m^3/t \); \( \eta_i \) is methane emission degree of the \( i \) adjacent strata, \%.  

It is noted that these methods are based on empirical correlations or standard assumed degree of emissions suitable for specific site conditions, errors may occur while applying the methods directly into other sites, where the mining methods and geological conditions may be quite different. Meanwhile, it is worth noting that the total methane emission of a longwall face can be estimated using these methods rather than the emission from individual sources.

Besides the above methods, Airey proposed a theory of gas emission prediction, through which the gas emission from each operation process can be determined, such as the emission from face rib, armoured face conveyor (AFC) and belt conveyor (Airey, 1971). In his theory, both the face advance rate and the abutment stress in front of face are taken into account by regarding the solid coal seam as an assembly of lumps or blocks of broken coal, with the size of the lumps depending on the their distance from the coal face. The emission characteristics from broken coal can be
derived from experimental tests conducted in the lab. Taking advantage of the integration method, the emission from individual sources thus can be determined. The mathematics of his theory is too lengthy to be presented here but some essential formulas proposed by Airey’s are given in Chapter 5 for the determination of methane emission from various operational processes, more information on the mathematical derivation process of his theory is detailed in Airey (1971).

2.2.2 Advanced prediction techniques based on numerical simulation

Due to the restriction of the empirical methods described above and the coefficients involved for specific site conditions, advanced reservoir simulations, interplaying the geomechanical and mining variables, are developed for methane emission prediction. Numerical reservoir simulations are capable to playback the complex interaction of mining induced rock failure and fracture, gas flow, and the influence of various gas drainage strategies on gas flow as well. They are flexible in adapting the models to different geological conditions, and optimising the drainage systems and mine designs accordingly (Kissell, 2006).

These methods also take the advantage of the ease of single parameter studies, e.g. by changing the value of one parameter and keeping the other parameters constant, the impact of this parameter can be achieved by a series of analysis, as a result, this is critical when field trials for single parameter study cannot be conducted.

In Australia, the computer based “Floorgas” and “Roofgas” simulation programs were employed to predict gassiness from floor and roof strata, design and optimise cross measure and in-seam drainage holes by indentifying stress relief areas up and down worked coal seam and degree of gas emission from these areas. It was reported that a high accuracy of gas emission prediction was achievable if sufficient geological, gas and mining data were provided (Lunarzewski, 1998).

A three-dimensional finite element model called Mine Gas Flow 3D (MGF-3D) was developed to predict the gas migration from mined coal seam and surrounding gas bearing rock strata on the basis of mining induced stress redistribution and
permeability changes (Tomita, et al., 2003). Meanwhile, a series of reservoir simulation based modelling has also been conducted by NIOSH, Karacan, et al. (2007) used a Generalized Equation-of-State Model (GEM) compositional reservoir simulator developed by Computer Modelling Group to estimate gas emission of a typical multi-panel Pittsburgh coaled mine. FLAC-2D (Itasca Consulting Group Inc., 2000) was involved in their approach to simulate the mechanical behaviour of rock in response to mining activities, through which the permeability of coal seam and rock strata after roof caving could be obtained and then incorporated into the GEM model.

With so many approaches available, engineers should be careful while making precise emission prediction, as some coefficients (degassing coefficient) developed for site specific conditions vary from site to site and are critical using traditional methods. For mining activity conducted with a deep burial depth where empirical data may not be suited or not available, numerical reservoir based technique seems to be a good choice. However, it should be noted that a detailed and correct survey or understanding of geological conditions such as rock properties and in-situ stress distribution is necessary for model construction and validation to gain a realistic reflection of mining process and mining induced issues.

2.3 CFD modelling of ventilation and gas flow in underground coal mines

The use of the CFD modelling technique has gained its popularity in almost every branch of fluid dynamics and engineering fields, ranging from aerodynamics of aircraft, automotive, pollution control, agriculture, food science, power plant, civil engineering, hydrology, oceanography, and medical science (Anderson, 1995; Versteeg and Malalasekera, 2007). There is no exception in mining industry where the ventilation system is essential for the dilution and removal of hazardous gas and dust. Since the early nineties of the 20th century, the CFD modelling approach has become an indispensable predictive and design tool in the mining industry. With the advancement of high performance computers, the trend of using CFD models to solve mine safety and health problems has increased dramatically.
In combination with experimental measurement, the applications of the CFD modelling technique in underground coal mines are numerous, including modelling of fundamental airflow patterns, investigation of goaf gas flow patterns, optimisation of methane drainage and goaf inertisation strategies, prediction of dust flow patterns and its control, evaluation of spontaneous combustion of coal in the goaf, coal mine fires and thermodynamic issues control. This section provides a critical review of the CFD investigations of the air and gas flow patterns in underground coal mine ventilation system.

Heerden and Sullivan (1993) were among the first who employed CFD models to investigate the airflow characteristics in underground headings, based on the modelled flow field, they also examined both the dust and methane flow behaviour around the continuous miner. However, they did not describe the validation of their models.

Srinivasa (1993) modelled the air velocities in a longwall face using a three dimensional CFD code. Due to limitations of computer resources at that time, only 40 m of the longwall face was modelled but it was reported that the model predicted results tallied well with field measurements.

Edwards, et al. (1995) claimed that there was great potential for using CFD models to solve mine safety and health related problems, such as the methane control, coal/gas outburst, dust suppression/diesel particulate matter, mine fires/explosions, spontaneous heating, heat and mine climate. A general process of conducting a CFD simulation was discussed, after which the simulation of ventilation patterns in a heading was presented as an example.

Oberholzer and Meyer (1995a, 1995b) used CFD models to evaluate the performance of different ventilation systems for the purpose of methane dilution and dust removal at development headings and they concluded that the CFD models were very effective in ventilation design.
Ren, et al. (1997) investigated methane flow and migration characteristics in the goaf and in particular around the longwall face. It is worth noting that the stress-permeability relationship obtained from laboratory tests was first incorporated in the CFD models to represent the mining induced permeability variations in the goaf.

Moloney and his colleagues (Moloney, 1997; Moloney, et al., 1999; Moloney and Lowndes, 1999) conducted CFD studies to investigate flow patterns within drivages of underground coal mines, where reasonable agreement with the results of physical model tests had been achieved.

Wala, et al. (1998) carried out CFD investigations to evaluate the performance of different ventilation systems in a heading face and demonstrated that the CFD technique could dramatically facilitate the design of ventilation systems. Later, Wala, et al. (2003, 2007) performed a series of benchmark experiments in a scaled physical model for the validation purpose of the CFD models. It is noticed that both airflow and methane concentration distribution in the heading were employed in the validation process. The impact of a dust scrubber on the air and methane flow behaviour in the heading was also examined, and Figure 2.3 illustrates the methane concentration distribution and airflow pathlines predicted in one scenario (Wala, et al., 2008)

![Figure 2.3 Methane concentration contour and airflow pathlines in a heading face](Wala, et al., 2008)
As indicated in Figure 2.4, Kelsey, et al. (2003) simulated the methane flow behaviour under different drainage practices, and they demonstrated that a combination of geotechnical modelling and CFD modelling could be used for the simulation of methane drainage. However, no calibration/validation of their CFD models was provided.

Balusu, et al. (2005b) developed longwall goaf models to investigate the goaf gas flow patterns and optimise the gas drainage strategies for highly gassy mines. It was reported that the results of the optimum gas drainage strategies were highly successful at the two underground coal mines that participated in the field demonstration.

Parra, et al. (2006) carried out both experimental tests and CFD studies to probe into the flow distribution in the heading of an underground coal mine. Upon validation of the model predicted flow field, they performed parametric studies to identify the distribution of dead zone and methane concentration within the heading under three different ventilation systems.
The modelling of ventilation flow in a heading with different cutting scenarios was reported by Hargreaves and Lowndes (2007). In their study, the simulated flow velocities were compared against the data obtained from ventilation experiments. They concluded that with the aid of CFD models, the planning and operation of auxiliary ventilation systems would be improved in terms of improving the dilution and removal of gas or dust emitted during the cutting operations. An example of the model predicted velocity vector distribution is shown in Figure 2.5.

Using 2D models, Aminossadati and Hooman (2008) examined the effects of brattice length on fluid flow behaviour in the crosscut regions.

![Velocity vector distribution in a heading (Hargreaves and Lowndes, 2007)](image)

Karacan, et al. (2008) discussed the advantages of the modelling work conducted by NIOSH in the U.S. and CSIRO in Australia. They concluded that the goaf gas flow patterns and gas distribution could be better understood through the grid-based numerical modelling studies, which had helped the development of effective goaf gas management strategies.

As shown in Figure 2.6, Toraño, et al. (2009) conducted CFD studies to investigate the methane flow behaviour in a development heading of an underground coal mine, where both airflow and methane concentration were measured at selected cross sections and points for the validation purpose. They demonstrated that the CFD modelling approach could be an effective tool for detecting zones in which methane tended to accumulate and thus requiring necessary ventilation enhancement.
Zheng and Tien (2009) simulated the methane flow behaviour on a longwall face considering the methane emission from various contributors, i.e., methane emitted from the coal broken by the shearer, the coal on the face conveyor, coal ribs and coal on the belt. The methane emission rate used in the model was derived from the field study conducted by Krog, et al. (2006). It is worth noting that they did not take into account the methane emitted from the goaf and the gas exchange between face and goaf was not considered either. The longwall model geometry employed in their study is illustrated in Figure 2.7.

![Figure 2.6 Methane concentration distribution on a cross section (Toraño, et al. 2009)](image)

Ren and Balusu (2010) summarised some typical applications of the CFD modelling technique in underground coal mines, including goaf gas management and drainage, goaf heating and inertisation, and longwall dust controls. Examples of the CFD applications on these specific aspects were provided; for instance, Figure 2.8 shows

![Figure 2.7 Geometry of longwall face model (Zheng and Tien, 2009)](image)
the CFD model they developed to investigate the goaf gas drainage efficiency using cross measure boreholes.

![CFD model diagram](image)

Figure 2.8 CFD modelling of roof boreholes intercepting seam gas from overlying seams (Ren and Balusu, 2010)

Ndenguma (2010) simulated the air and dust patterns in a heading using simplified CFD models, which were experimentally verified and validated using a scale down model.

Kenny, et al. (2012) applied the CFD modelling technique to understand both the air and methane flow patterns within a horseshoe heading under 11 different ventilation layouts, through which a good solution for methane dilution near the dead-end was obtained.

Kurnia, et al. (2012) and Guan, et al. (2012) investigated the thermal issue in the heading of underground coal mines. The performance of four turbulence models (Spallart-Almaras, $k-\varepsilon$, $k-\omega$ and RSM) were evaluated by comparing their results with the flow measurement conducted by Parra et al. (2006), and they found that the Spallart-Almaras turbulence model was the best to predict the flow behaviour. The impact of different ventilation layout and flow rates on the temperature distribution was examined.
Worrall (2012) modelled the flow of gases in longwall goafs with a focus on the distribution of explosive gas mixtures. Figure 2.9 illustrates the method he used to determine the range of explosive gas mixtures in the longwall goaf. On the basis of model results, suggestions were made to minimise the potential risk of longwall explosions.

Sasmito, et al. (2013) demonstrated that CFD modelling could be an effective approach to improve the ventilation system in underground coal mines. In their study, the airflow and methane distribution under different ventilation systems within the room and pillar mining entry and a cross-cut region were examined respectively. Figure 2.10 is an example showing the velocity field in the cross-cut region under different ventilation operations. It is interesting to notice that the validation was conducted on the turbulence models applied to the third computational model - a single end heading.

Torno, et al. (2013) studied the blasting gas behaviour in auxiliary ventilation of underground headings using both conventional and numerical models. Considering the gas dispersion with time, they developed the so called 4D CFD models. Validated by experimental measurements, they concluded that the CFD models could be used
to determine the time required to sufficiently dilute the hazardous gases for a safe working environment.

Figure 2.10 Velocity contours in the cross-cut region (Sasmito, et al., 2013)

From literature, it is known that the majority of these studies focus on the air and gas flow at development headings, except the work conducted by Srinivasa (1993) and Zheng and Tien (2009) who put their interest on the longwall face. And it is noticed that their models are either over-simplified or not properly illustrated. Therefore, to fill the gap, an important part of this thesis will be the modelling of ventilation and gas flow patterns along the longwall face, and more importantly to obtain fundamental understanding on the flow characteristics along face thus providing essential guidance to the face gas management.

2.4 Spontaneous combustion of coal and its control in underground coal mines

In this section, the occurrence of spontaneous combustion of coal is discussed first. Then the main research methods employed to understand this phenomenon are reviewed, in particular the applications of CFD models.

2.4.1 Spontaneous combustion of coal

Coal, once being exposed to the atmosphere, undergoes slow oxidation with the evolution of heat, gases and moisture. The heat produced in the oxidation process, if
not dissipated, increases the temperature of the coal, which further accelerates the rate of coal oxidation and heat release, and will eventually lead to an unstable situation and ignition of coal. This oxidation process is generally termed as spontaneous combustion or spontaneous heating or self heating.

As the spontaneous heating of coal produces toxic gases (like carbon monoxide) and has the potential of developing into mine fire, it has always been an issue of mine safety concern for mines operating seams liable to heating. In the U.S., approximately 17% of the 87 total reported fires for underground coal mines were caused by spontaneous combustion between 1990 and 1999 (DeRosa, 2004); whilst in Queensland, Australia, 51 spontaneous combustion incidents were identified between 1972 and 2004, three of which had led to mine explosions combined with 41 fatalities (Ham, 2005). Therefore, mining engineers and researchers have put persistent effort to understand the mechanism of self-heating of coal and the prevention of its further development into open fires.

2.4.2 Methods to investigate the spontaneous combustion of coal and its control

In underground coal mines, factors affecting the spontaneous combustion of coal can be categorised as self heating properties of the coal as well as the geological factors and the ventilation system associated with the mining method (Morris and Atkinson, 1986; Saghafi, et al., 1995). Generally, three methods are mostly adopted to investigate the impact of these factors on spontaneous combustion and this in turn contributes to the development of effective control measures.

Firstly, based on the adiabatic oxidation technique proposed by Davis and Byrne (1924), extensive laboratory tests have been conducted to evaluate the propensity of coal to spontaneous combustion. Using this technique, Humphreys (1979) defined a self-heating rate index $R_{70}$ to evaluate the tendency of Queensland coals to spontaneous combustion, and this index has been commonly used in Australia and New Zealand. Smith and Lazzara (1987) conducted measurements on U.S. coals and found that the self-heating tendency of a coal was highly affected by the humidity, the particle size and oxygen concentration of the air. Ren, et al. (1999) carried out
adiabatic oxidation tests to investigate the self-heating behaviour of pulverised coal samples. Throughout the tests, they identified two indexes (Initial Rate of Heating and Total Temperature Rise) that could be used to assess the liability of coal to spontaneous combustion; the influence of moisture on self-heating was also discussed. Beamish, et al. (2001) and Beamish (2005) performed adiabatic tests on Australian and New Zealand coal samples, and they found that the $R_{70}$ index was rank dependent. More recently, the relationship between $R_{70}$ and coal rank (together with the ash content and moisture) was further investigated by Beamish and Blazak (2005) and Beamish and Hamilton (2005).

Secondly, mathematical and numerical models have been developed to investigate the influence of factors affecting the self-heating process of coal and the inertisation of goaf. Arisoy and Akgün (1994) developed a mathematical model for spontaneous combustion, which was solved using the finite-difference technique. Through the model, they analysed the influence of gas velocity, particle size and inherent moisture content on the process of spontaneous heating. Saghafi, et al. (1995; Saghafi and Carras, 1997) modelled spontaneous combustion in the goaf of underground coal mines with a U ventilation system in two dimensions. Deng et al. (1999) proposed a mathematical model to calculate the temperature distribution in the goaf. Based on the model results, they put forward a minimum face advancing rate to avoid the occurrence of spontaneous combustion for Xinzhouyao Colliery, China. Investigations of the spontaneous heating and its control in the goaf are also conducted using CFD models which will be presented with more details in the following section.

Finally and the most importantly, field monitoring is the last effective method to detect the occurrence of spontaneous heating and evaluate the effectiveness of various heating control measures. One of the most commonly used goaf gas monitoring method is the tube bundle system, which allows a continuous drawing of gases from sealed off areas and an analysis of a wide range of heating gaseous products. The main limitation of this system is the travel time of the gas sample between entering a tube and analysis which can be controlled within acceptable levels. As a result, it has been widely accepted as an essential approach for
identifying the onset or development of spontaneous combustion (Vutukuri and Lama, 1986; McPherson, 2009). In conjunction with CFD studies, data obtained from the monitoring system had been used for the validation of base models and the evaluation of fast goaf inertisation strategies proposed from model results (Balusu et al., 2002; Ren and Balusu, 2009).

Together with the tube bundle system, inert gas, which was initially pumped into underground mines for fire fighting (Morris, 1987; Adamus, 2002), has become a major and common means for spontaneous combustion suppression (McPherson, 2009). Field practices also demonstrate the effectiveness and extension of inert gas from fire extinction to proactive goaf inertisation (Ren and Balusu, 2009; Donnelly and Bell, 2011). In addition to inert gas injection, slurry-grouting and gel infusion are also commonly adopted in mines for spontaneous combustion or fire suppression worldwide (Banerjee, 1985; Vutukuri and Lama, 1986; McPherson, 2009; Liang and Luo, 2008). It is worth noting that in China, the three-phase foam (composed of mud, nitrogen, and water) has been developed and widely used in underground mines for effective goaf inertisation and fire control (Wang, 2004; Zhou, et al., 2006).

2.4.3 CFD investigations of spontaneous combustion and its control

Besides the mathematical models, the CFD models are also employed to investigate spontaneous combustion in underground coal mines. Due to the sluggish air movement and oxygen penetration in the goaf, it has been identified as one of the most favourable places for the development of spontaneous combustion (Vutukuri and Lama, 1986; McPherson, 2009). Thus, the majority of CFD studies have focused on the simulation of gas and temperature distributions in the goaf as well as the development of various goaf inertisation strategies associated with its evaluation. A review of the literature can be summarised as:

Ren and Edwards (1997) developed CFD models to investigate the flow patterns in the goaf of an advancing face. From a perspective of spontaneous heating control, they put forward the potential of conducting nitrogen injection and its optimisation in the goaf.
Balusu, et al. (2002) modelled the gas distribution in longwall goafs when the face retreated near the finish-off line. Based on the model results, they conducted parametric studies and developed optimum goaf inertisation strategies aimed at achieving a fast goaf inertisation effect during the longwall sealing operation, which had been demonstrated to be highly successful. Figure 2.11 illustrates one of their goaf inertisation models when injecting inert gas behind a longwall face.

![Inert gas injection through 3 c/t seal](image)

Figure 2.11 Impact of inert gas injection on oxygen distribution in the goaf (Balusu, et al., 2002)

Lolon (2008) conducted CFD studies to detect hot spot locations in the longwall goaf, where the hot spot location was determined as a function of oxygen concentration and temperature, i.e., 5% for oxygen and 100°C for temperature. Figure 2.12 shows the temperature field in one modelled scenario. However, in his model, the goaf was divided into three zones where three constant values were assigned to these zones respectively representing the goaf permeability variation.

Taraba, et al. (2008) modelled the oxidation of coal and the corresponding evolution of generated heat and gases in the longwall goaf of a Czech mine; and they pointed out that an “optimal” zone for the development of spontaneous heating existed at a depth around 70 ± 20 m in the goaf under their specified condition. Furthermore, Taraba and Michalec (2011) examined the effect of face advance rate on oxidation heat production and evolution of gases in the goaf, and they found that spontaneous heatings could develop into flammable combustion if the face advance rate dropped
to a critical rate of 1 m/day. The temperature distribution in the goaf is presented in Figure 2.13 when the face advance rate is 1 m/day.

![Figure 2.12 Temperature field in one modelled scenario (Lolon, 2008)](image1)

Figure 2.12 Temperature field in one modelled scenario (Lolon, 2008)

![Figure 2.13 Temperature distribution in the goaf area with face advance rate at 1 m/day (Taraba and Michalec, 2011)](image2)

Figure 2.13 Temperature distribution in the goaf area with face advance rate at 1 m/day (Taraba and Michalec, 2011)

To investigate the effect of the ventilation scheme on the prevention of spontaneous combustion, Yuan et al. (2006) probed into the flow patterns within the goaf under three different ventilation systems using CFD models. Yuan and Smith (2007) further discussed the effects of the coal’s apparent activation energy and reaction surface area on the spontaneous heating process in the goaf. Later, Yuan and Smith (2008a) conducted transient modelling on the evolution of temperature field and oxygen distribution in the goaf under different ventilation conditions and goaf
permeability distributions. Meanwhile, they also studied the effect of coal properties on spontaneous heating in longwall goafs, and the oxygen concentration distribution in two adjacent goafs was predicted and is shown in Figure 2.14 (Yuan and Smith, 2008b); whilst Smith and Yuan (2008) simulated the spontaneous heating of coal in longwall goafs utilising a bleederless ventilation system. Following these studies, Yuan and Smith (2009) conducted 3D CFD modelling studies to simulate spontaneous heating in a large-scale coal chamber. They found that the calibrated CFD model was useful for predicting the induction time for spontaneous heating in underground coal mines. More recently, Yuan and Smith (2010) investigated the effect of face advance on spontaneous heating in longwall goafs, where the maximum temperature developed during the face stoppage was examined. It is noted that in their study the face advance was simulated as a series of discrete movements rather than continuous motion. In the meantime, Smith and Yuan (2010) modelled the impact of seal leakage on spontaneous heating in a longwall goaf which was ventilated by a Y type bleederless ventilation system. They found that the leakage rate and the goaf permeability had a major effect on the spontaneous heating process. However, in both studies, no data was available to validate the model predicted temperature field.

![Figure 2.14 Oxygen contour in longwall goafs (Yuan and Smith, 2008b)](image)

Trevits, et al. (2010) used CFD models to estimate the time required for different inert gas injection strategies to convert sealed areas into an inert atmosphere. Both injection rates and injection positions were considered in their study.
It can be seen from these references that the majority of these CFD studies are carried out in a single goaf or two adjacent goafs, and they focused on either the oxygen or temperature distribution or both of them; however, little attention was paid to the changing elevations of goaf which significantly affects the gas flow behaviour owing to buoyancy effects, and to the flow patterns of the gaseous products of spontaneous heating, which is vital for the early detection of spontaneous heating in terms of monitoring points arrangement. Consequently, one chapter of the thesis will concentrate on the modelling of gas flow behaviour in multi-goaf areas together with the impact of various operating conditions on the gas distribution considering the impact of goaf elevation variation.

2.5 Dust issue and its control in underground coal mines

2.5.1 Dust issue in underground coal mines

Dust contamination in underground coal mining ventilation systems is a common issue for mine operators. The generation and subsequent dispersion of coal dust into the mine ventilation system can not only be a health risk but also potentially contribute to coal mine gas explosion hazards. Unlike a methane explosion, dust kills miners in a long term, and the time may vary from a few weeks to ten or twenty years, depending on the mass concentration and composition of dust particles (Colinet, et al., 2010). Fortunately, miners’ awareness of the dangers is now recognised and when working in heavy concentrations of dust improved working conditions are always demanded. Researchers have been working on measurement and development of respirable dust control strategies (Kissell, 2003; Volkwein, et al., 2004a and 2004b; Gillies and Wu, 2005; Gillies and Wu, 2006; Ren and Balusu, 2007; Xie, et al., 2007; McPherson, 2009; Colinet, et al., 2010; Ren, et al., 2011). Governments also promulgate laws to ensure a better working environment is provided for miners (Anon., 2006). However, for various reasons, management of respirable dust in underground coal mines remains a challenging issue for mine operators and some coal mines still cannot meet the requirement (Rider and Colinet,
2007; Plush, et al., 2011). Thus, a better understanding of the aerodynamics of dust particles is in great demand.

2.5.2 Approaches to investigate the dust flow behaviour

Three approaches are generally used to investigate the dust behaviour in the mining industry, namely, field monitoring, mathematical models and computational models.

Methods of measuring and sampling respirable dust particles are quite numerous but can be classified into several distinct categories, including the particle count methods, gravimetric methods, photometric (light-scattering) methods, and personal samplers (McPherson, 2009). Except the particle count methods, the other three methods are now commonly used for dust evaluation in underground coal mines. Each of these methods has its own pros and cons. The gravimetric methods are typically used for long-term (usually a work shift) time-weighted average dust measurement either for personal sampling or area sampling, however, instant reading is not available. In New South Wales, Australia, the personal gravimetric sampling method is adopted and legislated to evaluate the operators’ dust exposure level (Plush, et al., 2012). The photometric dust instruments were developed on the basis of scattering of light by dust particles, which can be employed for short term and long term sampling with immediate or any given time interval readings provided. NIOSH has completed the pioneer work on personal samplers which are able to monitor temporal variation of dust concentrations to which the operators are subjected. The monitor was further modified to be person-wearable Personal Dust Monitor (PDM) can accurately predict a miner's dust exposure (Volkwein, et al., 2004a; 2004b). This equipment has also been used in Australian coal mines for real time dust monitoring at development headings and longwall faces (Gillies and Wu, 2005; 2006).

Hwang et al. (1974) developed a mathematical model using an Eulerian approach, which was capable of modelling four different types of dust source: point source, line source, moving line source and flat plane. However, no comparisons with field data or experimental results were made at that stage. Courtney et al. (1986) proposed models to investigate deposition of respirable dust for underground coal mines, and
found that the respirable dust deposition rate decreased as a function of distance from the source. Taking into account the convective diffusion, gravity, coagulation, collision mechanisms and re-entrainment of particles, Bhaskar and Ramani (1988) proposed another dust deposition model, which was applicable to various dust particle sizes. Using a statistical method, the model developed by Chiang and Peng (1990) was able to predict the dust concentration distributions along a longwall face with satisfactory agreement with field data. It is noted that these mathematical models use either Eulerian or Gaussian algorithms.

2.5.3 General practices for dust control

Practices for dust control in coal mines are quite site specific but can be classified into several distinct categories, including suppression by water infusion or wet cutting methods, dilution by ventilation system, mitigation by water sprays and scrubbers, and isolation by personal masks (McPherson, 2009). Suppression is the most effective method used for respirable dust control in terms of reducing the amount of respirable dust particles. It refers to methods used to prevent the dust particles from becoming airborne and it is usually achieved by pre-wetting of coal and wet cutting methods.

Dust particles with a larger diameter are more likely to settle out by gravitational sedimentation; whilst for respirable dust particles, they tend to suspend in the roadway and travel with the airstream. Therefore, besides providing fresh air to the work force and diluting the gas, another important function of mine ventilation is to dilute the concentration of dust particles suspended in the air. However, it does not mean higher air quantities can result in lower dust concentration especially when sufficient moisture is not present in the working environment. Previous study on the relationship between air velocity and dust concentration indicated that the settled dust particles might become airborne and re-enter the roadway causing secondary contamination to the airstream (Vutukuri and Lama, 1986; Colinet, et al., 2010). A typical curve showing the relationship between air velocity and dust concentration of different dust sizes in roadways is depicted in Figure 2.15.
In most cases, ventilation dilution is not adequate to guarantee the respirable dust concentration is under the statutory limit. Therefore, various dust mitigation systems have emerged. Among these systems, water sprays are widely employed for dust mitigation at locations such as longwall faces, headings, coal crushing and transfer points. Depending on the specific application, several types of water sprays are available for use at these locations (Colinet, et al., 2010). For example, hollow-cone sprays are mostly located close to dust source for suppression, while full-cone sprays are more used further away from the dust source or can be used for wetting of coal at transfer points. Although air-atomising sprays are reported to be most effective in airborne dust capture, their utilisation has been limited due to high maintenance requirements (Kissell, 2003; Colinet, et al., 2010). Besides the water sprays, shearer clearers or scrubbers are becoming popular in mining applications due to their high respirable dust capture efficiency especially when massive dust particles are generated around shearer and continuous miner (Kissell, 2003; Ren and Balusu, 2007; Aziz, et al., 2009).

Another technique that can be used for dust mitigation is foam. In the 1980s, Laurito and Singh (1987) had conducted field tests to evaluate the performance of foam on dust mitigation at longwall faces. Their results indicated that 50-70% of dust the shearer operators were subjected to could be reduced by discharging foam from shearer drum (Colinet, et al., 2010). Ren (2009) developed a foam agent and foam maker that could be used for underground coal mines. Field application at a development heading also demonstrated that a promising dust mitigation effect could
be achieved using the foam technique. Engineers are also trying to minimise operators dust exposure level by applying a uni-directional cutting system at longwall faces so that operators do not need to work in the downwind of dust-laden air (Aziz, et al., 2009).

Besides the traditional methods for dust control, a water-mist based venturi system which could be used for dust control at maingate chocks and Beam Stage Loader (BSL) had been developed by Ren and his colleagues (Ren, et al., 2011). Being able to produce ultra fine water droplets, this system is particularly effective for the capture of respirable dust particles, and field trials conducted at longwall faces of Australian coal mines demonstrated that up to 35% of the respirable dust would be removed from the atmosphere (Ren, et al., 2012). In South Africa, a fogger system using water mist technology has been investigated at Thandeka and Twistdraai as water curtains on the intake airway and transfer points (Ren, et al., 2011). Field results proved that the system was highly effective by reducing dust concentrations by 96% during the test periods. More information on the water-mist based venturi system and field trials of this new technique will be presented in Chapter 8.

2.5.4 CFD modelling of dust flow behaviour

Compared with the modelling of air and gas flow patterns in underground coal mines, the CFD studies of the dust flow behaviour are relatively weak and further studies need to be conducted to gain a better understanding of the dynamic dispersion of coal particles.

The early CFD studies of dust have focused on the modelling of dust distribution and its suppression in development headings (Sullivan and Heerden, 1993a and 1993b; Oberholzer and Meyer, 1995b), while the simulation of dust concentration distribution at longwall faces has only been reported by Srinivasa, et al. (1993). After 2000, there has been a slight increase in investigating the dust flow dynamics driven by the increasing demand of healthy working environments.
Balusu et al. (2005a) investigated the air and respirable dust flow patterns around the longwall shearer, and evaluated the effect of various parameters and dust control options. Figure 2.16 is a snapshot showing the dispersion of respirable dust particles near the longwall shearer.

![Figure 2.16 Respirable dust flow patterns near the shearer (Balusu et al., 2005a)](image)

Ren and Balusu (2008; 2010) modelled the dust flow patterns around the longwall shearer and walkway under different operating conditions based on CFD models, where a shearer scrubber was incorporated into the models to evaluate its performance on dust suppression before implementing field trials. Figure 2.17 illustrates the impact of the scrubber on dust flow behaviour near the shearer with the scrubber inlet facing the shearer drum.

![Figure 2.17 Impact of shearer scrubber on dust flow behaviour (Ren and Balusu, 2010)](image)
Toraño et al. (2011) analysed dust flow behaviour in two auxiliary ventilation systems by CFD models, where both airflow velocity and respirable dust concentration profiles were used to assess the accuracy of model results. They predicted the dust evolution in the roadway and pointed out that dust concentration at some points may exceed the statutory limit, corresponding modifications to the ventilation scheme were also suggested. Consequently, they concluded that the CFD models could be used to optimise the auxiliary ventilation system for better dust removal at headings. The dust concentration distribution in the two models can be found in Figure 2.18.

![Figure 2.18 Dust concentration isocontours in mg/m³ on the cross section located 0.5 m from heading (Toraño, et al., 2011)](image)

It is known from the literature that the CFD investigations of dust flow behaviour in underground coal mines are still limited to the heading and longwall shearer, indicating inadequate attention has been paid to dust contamination at the longwall entrance and in the vicinity of underground bins where the dust issue has been raised by field surveys or observations (Gillies and Wu, 2007; 2008). Regarding the dust issue above the underground bin, Silvester et al. (2004; 2005; 2007) simulated the airflow and dust dispersion patterns above the crusher feed bin of an underground metal mine, which is different with the bin of underground coal mines in terms of ventilation system, feeding method and site configurations. Therefore, the modelling of dust flow behaviour at the longwall entrance and in the vicinity of underground bins for underground coal mines is of great significance for the development of an effective dust mitigation system. The corresponding studies related to the two aspects are described in Chapters 8 and 9 respectively.
2.6 Summary

A comprehensive review of the methane emission prediction at longwall faces, spontaneous combustion of coal and the fugitive dust dispersion in underground coal mines has been carried out in this chapter, with a particular focus on the application of CFD models to solve these issues. Among the various methane emission prediction methods developed worldwide, Airey’s mathematical model was found to be superior in determining the methane emission from different sources at longwall faces, and could be further used as essential input parameters for the modelling of longwall methane flow patterns. Meanwhile, review on the air and gas flow modelling in underground ventilation systems has demonstrated the necessity of simulating the air and gas flow characteristics at longwall faces. Then, the occurrence of spontaneous combustion and its control were discussed, through which the limitations of current CFD investigations were identified. Furthermore, research on the dust issue reported in existing publications was reviewed, indicating the great demand for dust flow modelling at longwall entrance and in the vicinity of underground bin as well as the development of corresponding dust mitigation strategies. To summarise, this comprehensive literature review opens up the way to carry out further innovative CFD investigations of the gas (air and multi-species gas mixture) and dust flow behaviour in underground coal mines in the following chapters.
3 INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS

3.1 Introduction

The equations of fluid mechanics, which have been known for over a century, were not solvable in general until the development of computer technology. Ever since the power of the computers was recognised, applications of numerical techniques to investigate fluid flow behaviour increased dramatically, leading to the development of a new branch of fluid dynamics – Computational Fluid Dynamics (CFD). Nowadays, with the availability of high performance computers, the CFD technique has developed into a new stage and is broadly adopted in almost every branch of fluid dynamics and the engineering field, and the mining industry is no exception. This chapter provides a brief background of the CFD technique that is used in subsequent chapters to model the flow behaviour of hazardous gas and dust. As the CFD code employed in this thesis is the ANSYS Fluent (ANSYS, 2010), reference is made to this code when necessary.

3.2 Governing equations

Fluid (gas and liquid) flows are governed by partial differential equations which represent conservation laws for mass, momentum, and energy. CFD is the art of replacing these partial differential equations with a set of algebraic equations (the process is called discretisation), which in turn are solved with the aid of a digital computer to obtain an approximate solution (Anderson, 1995). Therefore, it is imperative to understand the partial differential equations governing the fluid flow before introducing the numerical solution methods which have been developed to solve them.
3.2.1 Continuity equation

The continuity equation is based on the principle of mass conservation, which means the rate of increase of mass in the fluid element must be equal to the net rate of flow of mass into the fluid element. For a Newtonian fluid, such as air, the mass conservation equation in finite volume can be given in the following form by using Reynolds transport theorem:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$  \hspace{1cm} (3.1)

where $\rho$ is the fluid density, and the first term $\partial \rho / \partial t$ represents the variation of fluid density with time; $\mathbf{u}$ is the velocity vector, and the term $\nabla \cdot (\rho \mathbf{u})$ accounts for the net flow of mass out of the element across its boundaries and is called the convective term.

For incompressible fluid flows where the density is constant, the continuity equation becomes:

$$\nabla \cdot (\rho \mathbf{u}) = 0$$  \hspace{1cm} (3.2)

Or in three dimensions,

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$  \hspace{1cm} (3.3)

3.2.2 Momentum equation

The momentum equation follows Newton’s second law, which states that the rate of change in momentum of a fluid particle is equal to the sum of forces acting on it. As a result, the principle of conservation of linear momentum (i.e. the well-known Navier-Stokes equation) is expressed by:
\[
\rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \mu \nabla^2 \mathbf{u} + \mathbf{F} \tag{3.4}
\]

where \( \rho \) is the fluid density, \( \mathbf{u} \) is the velocity vector of fluid particle, \( \nabla p \) is the pressure gradient, \( \mu \) is viscosity of fluid, and \( \mathbf{F} \) is the body force vector.

The terms \( \rho \frac{\partial \mathbf{u}}{\partial t} \) and \( \mathbf{u} \cdot \nabla \mathbf{u} \) on the left hand side of Eq. 3.4 represent force components caused by the rate of momentum change and convective acceleration, respectively. These two terms together represent force from inertial effect, while the term on the right hand side of Eq. 3.4 \( \mu \nabla^2 \mathbf{u} \) represents viscous force.

### 3.2.3 Energy equation

The first law of thermodynamics, which states that the rate of change of energy of a fluid particle is equal to the rate of heat addition to the fluid particle plus the rate of work done on the fluid particle, composes the energy equation that the fluid flow needs to follow. Thus, the principle of energy conservation can be given by:

\[
\rho \left( \frac{\partial E}{\partial t} + \mathbf{u} \cdot \nabla E \right) = K \nabla^2 T + W_s + S_E \tag{3.5}
\]

where \( E \) is energy, \( K \) is conductivity, \( W_s \) is work done by the surface stresses and \( S_E \) is the energy source term.

As a result, when the flow is compressible or there is a temperature gradient in the flow domain, the energy equation must be solved.

### 3.2.4 Equations of state

So far, it is known that fluid motion in three dimensions is described by five partial differential equations for mass, momentum (in three dimensions), and energy. However, there are a total of seven unknown variables, causing closure problem for
the governing equations. It is noticed that four of the unknowns are thermodynamic variables: $\rho$, $p$, $i$ (internal/thermal energy) and $T$. By assuming the thermodynamic equilibrium, relationships between these variables can be obtained, which are referred to as equations of state and can be written as:

$$p = p(\rho, T) \quad \text{and} \quad i = i(\rho, T) \quad (3.6)$$

Specifically, for a perfect gas, equations of state can be expressed as:

$$p = \rho RT \quad \text{and} \quad i = C_v T \quad (3.7)$$

where $R$ is the gas constant and $C_v$ is the specific heat of constant volume.

Now, it can be stated that the equation system is mathematically closed by a combination of the five partial differential equations with two additional algebraic equations.

3.2.5 Generic transport equations

It is worth noting that there are significant commonalities between the various equations and all the dependent variables seem to obey a generalised conservation principle. If a general variable is denoted by $\phi$, then the generic differential equation can be written in the following form:

$$\frac{\partial \rho \phi}{\partial t} + \nabla \cdot (\rho \phi \mathbf{u}) = \nabla \cdot (\Gamma \nabla \phi) + S_\phi \quad (3.8)$$

where $\Gamma$ is diffusion coefficient.
The various physical transport processes occurring in the fluid flow are clearly highlighted by Eq. 3.8: the transient term, \( \frac{\partial \rho \phi}{\partial t} \), indicates the rate of change of \( \phi \) in the concerned control volume; the convection term, \( \nabla \cdot (\rho \phi \mathbf{u}) \), stands for the transport of \( \phi \) due to the existence of the velocity field; the diffusion term, \( \nabla \cdot (\Gamma \nabla \phi) \), accounts for the transport of \( \phi \) due to its gradients; and the source term, \( S_\phi \), represents any sources or sinks that either create or destroy \( \phi \).

3.3 Discretisation method

Due to the non linearity of the partial differential equations, direct solutions are not available. However, an approximate solution can be obtained numerically by adopting a discretisation method, through which the governing equations are approximated by a system of algebraic equations for the variables at some set of discrete locations in space and time (Ferziger and Peric, 2002). Many discretisation methods have been developed so far, but the most commonly known are: Finite Difference Method (FDM), Finite Volume Method (FVM) and Finite Element Method (FEM).

Among these discretisation methods, the FDM is the oldest for the numerical solution of partial differential equations and it uses the differential form of conservation equations as its starting point. It is simple and effective on structured grids where it is also easy to obtain higher order differencing approximation. Typical examples of one-dimensional and two-dimensional Cartesian grids used in the FDM are illustrated in Figure 3.1 (Tu, et al., 2007). However, the main disadvantage of the FDM is that the conservation is not always enforced (Tu, et al., 2007). Furthermore, the restriction to simple geometries with highly structured grids composes another significant disadvantage of the FDM.

The FVM discretises the integral form of the conservation equations directly in physical space. Using FVM, the solution domain is subdivided into a finite number of contiguous control volumes (cells) by a grid (as indicated in Figure 3.2). Then
integration is carried out over each control volume (CV) and by applying Gauss’ divergence theorem, the integral form of governing equations can be expressed as:

$$\frac{\partial}{\partial t} \left( \int_{CV} \rho \phi dV \right) + \int_{A} \mathbf{n} \cdot (\rho \phi \mathbf{u}) dA = \int_{A} \mathbf{n} \cdot (\Gamma \nabla \phi) dA + \int_{CV} S_{d} dV \quad (3.9)$$

where \( \mathbf{n} \) is the vector normal to surface element \( dA \).

As a result, Eq. 3.9 constitutes the actual form of the conservation equations solved by finite volume based CFD programs to calculate the flow pattern and associated scalar fields.

Figure 3.1 One-dimensional and two-dimensional uniformly distributed Cartesian grids for the FDM (full symbols denote boundary nodes and open symbols denote computational nodes) (Tu, et al., 2007)

As the computational node lies in the centroid of each CV rather than the grid intersection point, the FVM has the capacity to accommodate any type of grid, which offers greater flexibility for dealing with complex geometries. This may contribute to the adoption of FVM in almost all commercial CFD codes to obtain numerical solutions for complex fluid flow problems.
Figure 3.2 A representation of structured and unstructured mesh for the FVM (full symbols denote element vertices and open symbols at the centre of the control volumes denote computational nodes) (Tu, et al., 2007)

Regarding the FEM, it is similar to FVM in many aspects, except that the equations are multiplied by a weight function before they are integrated over the entire domain (Ferziger and Peric, 2002). More details about the FEM can be found in Baker (1983).

3.4 Numerical method

The numerical solutions to the system of algebraic equations can be achieved by either direct methods or iterative methods. However, the cost of using direct methods is generally quite high owing to the large system of non-linear equations obtained in majority of CFD problems. Thus, the indirect iterative methods are predominantly used in CFD solvers.

It is noted that for incompressible flows, the solution of governing equations is complicated by the lack of an independent transport equation for pressure. Therefore, various schemes have been developed to link the pressure with the velocity for
incompressible flow problems, and one of the most broadly adopted is the SIMPLE scheme, which stands for Semi-Implicit Method for Pressure-Linkage Equations. The SIMPLE scheme was originally developed by Patankar and Spalding (1972) and it adopts a guess-and-correct procedure for the calculation of pressure and velocity. The solution procedure for the SIMPLE algorithm is described in Figure 3.3. It can be seen that, firstly, a pressure field is guessed to initiate the SIMPLE calculation process, based on which a velocity field can be calculated through the discretised momentum equations. Then, a pressure correction equation, derived from the continuity equation, is solved to obtain a pressure correction field, which is subsequently used to correct the velocity and pressure fields. These guessed fields are continuously improved in an iterative method until convergence is achieved.

The use of the SIMPLE scheme requires the under relaxation in the iterative process, which controls the stability of calculation. Thus, the relaxation factor $\alpha$ is introduced in the calculation as follows:

$$\phi_{n+1} = \phi_n + \alpha \Delta \phi$$  \hspace{1cm} (3.10)

Figure 3.3 The SIMPLE algorithm solution procedure
where $n$ represents the number of iteration and $\Delta \phi$ accounts for the calculated variation of $\phi$ within a given iteration.

Generally, when $\alpha < 1$, it is termed as under relaxation, which increases the stability of calculation at the cost of lowering the speed of convergence; when $\alpha > 1$, it is referred to as over relaxation, which can be used to accelerate the convergence speed but the stability will be greatly weakened; and no relaxation takes place when $\alpha = 1$.

Another issue that needs to be mentioned in the iterative methods is the convergence criteria. The residuals, which measure the imbalance in conservation equations, are usually monitored to evaluate the convergence of the solution during the numerical procedure. For most applications, a numerical solution can be thought converged when the scaled residuals fall in the range of 1E-3 to 1E-4 or less. It is noted that the residuals cannot reach zero due to the inherent errors and approximations adopted in the calculation. Meanwhile, specific variables can be monitored to further determine the convergence of a solution under certain circumstances. A typical convergence history during a calculation can be found in Figure 3.4.

![Figure 3.4 A typical convergence history during a calculation](image-url)
3.5 Mesh generation

The generation of a mesh is one of the most important steps in the pre-process of any CFD problems. As described before, upon defining the computational geometry, the domain needs to be subdivided into a number of small control volumes (take the FVM as example) in which the discretised governing equations are solved. The quality of mesh is important as it significantly affects the rate of convergence, solution accuracy and the computational time. A high quality of mesh is always desired to obtain an accurate numerical solution within reasonable time. Generally, it is recommended that the mesh density should be higher in regions where sharp gradients in flow variables are present.

Two types of mesh are generally adopted: structured mesh and unstructured mesh. The structured mesh refers to a single type of element in a regular arrangement, while the unstructured mesh allows arbitrary combinations of hexahedral, tetrahedral and prism elements. Both types of mesh have their own advantages and disadvantages. Generally, with the same cell count, hexahedral meshes will give more accurate solutions, especially when the grid lines are aligned with the flow. However, the application of structured meshes is primarily limited to simple geometries. By contrast, the unstructured meshes have great flexibility to deal with flows within complex geometries. Therefore, for the models of longwall face and underground bin developed in subsequent chapters, typical unstructured meshes are adopted to accommodate the complex geometries encountered in the underground environment.

In ANSYS Fluent, three main indicators are commonly used to evaluate the quality of meshes: orthogonal quality, aspect ratio and skewness (ANSYS, 2010). Specifically, orthogonal quality is computed for cells using the vector from the cell centroid to each of its faces, the corresponding face area vector, and the vector from the cell centroid to the centroids of each of the adjacent cells (ANSYS, 2010). According to its definition, the orthogonal quality varies from 0 to 1; and the closer this value to 0, the worse the cell. Aspect ratio is a measure of the stretching of the cell which can be simply calculated as the ratio of the longest edge length to the
shortest edge length. Generally, sudden changes of aspect ratio should be avoided, especially in areas where the flow exhibits large changes or strong gradients (ANSYS, 2010). Skewness is defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Efforts should be made to ensure the maximum skewness of a mesh is lower than 0.95.

### 3.6 Boundary conditions

Boundary conditions, which are essential to solve the governing equations, consist of flow inlets and exit boundaries, wall, repeating, pole boundaries and internal face boundaries. For different flow problems, the application of boundary conditions varies. Regarding the gas and dust related calculations conducted in the thesis, three types of boundary conditions are mostly applied: inlet, outlet and wall. And this section presents a brief description of these boundary conditions employed in the thesis.

#### 3.6.1 Inlet boundary conditions

Inlet boundary conditions define the entrance of flow into the solution domain. A wide range of inlet types are available in ANSYS Fluent, including velocity inlet, pressure inlet, mass flow inlet, inlet vent/intake fan (ANSYS, 2010). Specifically, their definitions and applicable conditions are as follows (ANSYS, 2010):

- **Velocity inlet boundary conditions** are used to define velocity vector and scalar properties of flow at inlet boundaries. It is intended for incompressible flows and is useful when velocity profile is known at inlets.
- **Pressure inlet boundary conditions** define the total gauge pressure, temperature, and other scalar quantities at flow inlets, which can be applied to both incompressible and compressible flows. However, the flow direction must be given when using this boundary condition.
- **Mass flow inlet boundary conditions** are used in compressible flows to prescribe a mass flow rate at an inlet. For incompressible flows where fluid
density is constant, the mass flow is fixed with velocity inlet boundary conditions.

- Inlet vent/intake fan are used to model inlet vent/external intake fan with specified loss coefficient/pressure jump, flow direction, and ambient (inlet) pressure and temperature.

Finally, velocity inlet boundary conditions are used in the model calculations to permit the flow entering the interested working environment.

### 3.6.2 Outlet boundary conditions

Corresponding to the various inlet boundary conditions, outflow, pressure outlet, pressure far-field, and outlet vent/exhaust fan compose the diversity of outlet boundary conditions (ANSYS, 2010). The characteristics of these outlet boundary conditions can be summarised as follows (ANSYS, 2010):

- Outflow boundary conditions are particularly used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. However, they cannot be used with compressible flows as well as with pressure inlet boundary conditions. When back flow occurs at the exit, it is not appropriate to use outflow boundary conditions.
- Pressure outlet boundary conditions are used to define the static pressure of the environment into which the flow exhausts. Compared with outflow boundary conditions, back flow can occur at pressure outlet boundaries and the use of pressure outlet boundary conditions can minimise convergence difficulties thus often resulting in a better rate of convergence.
- Pressure far-field boundary conditions, which are available only for compressible flows, are usually used to model a free-stream compressible flow at infinity, with free-stream Mach number and static conditions specified.
- Outlet vent/exhaust fan are used to model external outlet vent/exhaust fan with specified loss coefficient/pressure jump and ambient (discharge) pressure and temperature.
As a result, the pressure outlet boundary conditions are broadly adopted in the simulations conducted in this thesis.

3.6.3 Wall boundary conditions

Upon defining the inlet and outlet boundary conditions, the wall, which bounds the fluid and solid regions, is the most common boundary encountered in confined fluid flow problems. In viscous flows, the no-slip boundary condition is the appropriate condition for the velocity at solid walls. Motions can be added to the wall to generate the moving wall boundary conditions with respect to the stationary wall boundary conditions. Meanwhile, the moving boundary conditions can be specified as either translational or rotational walls in terms of movement mode. In addition, thermal, species and discrete phase boundary conditions are also applicable in defining the wall boundary conditions for desired heat transfer, species transport and discrete phase calculations respectively.

3.7 Post processing

When convergence is reached for a particular simulation, post processing can be initiated to visualise and analyse the modelled results. The majority of CFD codes embed post processing functions into their main codes, and there are also many stand alone softwares developed in particular for the post processing of CFD data. There is no criterion in terms of determining which tool should be used, it is user preference dependent. Regardless of the tools used for post processing, the main aim of post processing is to get the most out of the model predicted results and to provide complete insights into the fluid flows investigated. In this thesis, the ANSYS Fluent was also used as the post processing tool to interpret the simulated results (ANSYS, 2010).

ANSYS Fluent provides a number of approaches to interpret the CFD results, i.e., flow variables can be presented in terms of contours, vectors, plots, iso-surfaces, surface integrals, and so forth. Meanwhile, a wide range of flow variables are also
available in ANSYS Fluent which indicates the abundant data provided by the CFD simulation. In addition, custom field functions are allowed to define variables according to users’ needs. Therefore, aided by the colour and light effects, not only a detailed and colourful but also meaningful flow field visualisation can be obtained in the entire domain as well as at any position within it, thus providing a thorough understanding of complex flow phenomena. Through these visuals, issues encountered in the fluid flows can be easily identified, which is favourable to the minimisation and elimination of undesired phenomena.

### 3.8 Errors and uncertainties

As described before, the equations governing fluid flow and its related process can be solved approximately by the CFD approach whereas exact solutions are still not available. Currently, it is generally accepted that the discrepancy between the approximate solution and the exact solution falls into two categories: errors and uncertainties (Versteeg and Malalasekera, 2007). And the following definitions have been widely accepted: error is defined as a recognisable deficiency in a CFD model that is not due to lack of knowledge whereas uncertainty is defined as a potential deficiency in a CFD model that is due to lack of knowledge (AIAA, 1998; Oberkampf and Trucano, 2002). According to the definitions, causes of errors include numerical errors, coding errors and user errors, and the numerical errors can be subdivided into roundoff errors, iteration errors and discretisation errors; whilst the main sources of uncertainty involve input uncertainty and physical model uncertainty, and the input uncertainty can be attributed to the simplified domain geometry, inaccurate implementation of boundary conditions and the assumption of fluid properties. A detailed description on the components of errors and uncertainties can be found in Versteeg and Malalasekera (2007).

Upon recognising the unavoidable errors and uncertainties in CFD modelling, rigorous methods have been developed to assess the errors and uncertainties encountered in the model results, which are termed as verification and validation. In the context of CFD modelling, the broadly accepted definitions of verification and validation are: verification is the process of determining that a model implementation
accurately represents the developer’s conceptual description of the model and the solution to the model; while validation is the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model (AIAA, 1998; Oberkampf and Trucano, 2002).

Obviously, the verification process quantifies errors whilst the validation process quantifies the uncertainties. Generally, the verification has been conducted by the CFD code providers as part of their development; meanwhile, according to the users’ requirement, the verification may include the implementation of different precision levels, mesh refinement, discretisation schemes and so forth. The common practice of the validation process is the comparison of model results with experimental or analytical results. However, in the mining industry, due to the high cost and safety issues of an experimental setup, model validation is generally achieved by comparing against field monitored data though some simplified experimental tests that have been conducted by some researcher (Ren and Balusu, 2010; Wala, et al., 2007).

Once finishing the verification and validation process at desired levels, the users will be more confident to trust the simulated results. Then a comprehensive post processing and analysis can be carried out to gain insightful solutions to the fluid flow and its related process.

3.9 Summary

The main purpose of this chapter is to provide a brief introduction to the principles of the CFD modelling technique. CFD is the art of solving the governing equations numerically by adopting various approximation methods to infinitely approach to the exact solution of the flow phenomena with the aid of digital computers. In this chapter, the equations governing the fluid flow are introduced first, followed by the discretisation and numerical methods which have been developed to solve them; then the requirement of mesh and boundary conditions in the solution domain are discussed; finally, the post processing and the errors and uncertainties of model results are illustrated, as well as the necessity of verification and validation of model results to quantify the unavoidable errors and uncertainties. Bearing in mind these
principles, the CFD modelling technique has been successfully used in subsequent chapters to model the flow behaviour of hazardous gas and dust encountered in the underground coal mining industry.
4 MODELLING OF AIR FLOW PATTERNS ON A LONGWALL FACE

4.1 Introduction

The longwall face is the main production area of a modern underground coal mine, where more than 90% of coal production is achieved. As a result, the longwall face becomes the largest gas and dust contributor of the mine, and the place where majority of disasters occurred. Thus, ventilation to the longwall face is vital in terms of diluting the hazardous gas and dust to ensure a safe and healthy working environment.

Mining engineers have always tried to understand the ventilation of longwall faces. The most common way is to monitor the flow velocity and the pressure drop at different cross sections along the longwall face. Instruments that could be used to measure the velocity have been developed from the vane anemometers to the more convenient digital anemometers which can be used for point velocity measurement with acceptable accuracy. However, these instruments are not able to capture the turbulence nature of flow field on a longwall face, and some areas are not reachable to carry out the measurement due to the complex configurations of the longwall face. With the use of the CFD modelling technique, there will be no such disadvantages and a detailed airflow field can be predicted not only along the walkway but also around the shearer on the face. Assisted by the field measured velocity profile along face, the validation of base model results will constitute a prerequisite to investigate the airflow characteristics using CFD models.

This chapter describes the development of CFD models for investigating the flow characteristics on a typical longwall face. Parametric studies are also conducted to understand the impact of a series of operational scenarios on the flow patterns following the validation of the base CFD model.
4.2 Development of longwall CFD models

An actual longwall face configuration can be very complex due to the appearance of chocks, shearer, Armoured Face Conveyor (AFC) and electric cables within the constrained longwall face space. There is no doubt that the complex longwall geometry results in a complex flow field along the face. Numerical investigations of the flow characteristics of longwall faces have been conducted by Srinivasa (1993) and Zheng and Tien (2009), however, their models only include part of the longwall face or the geometry of the models are not sufficiently illustrated, a detailed airflow field or some important characteristics of the flow field cannot be obtained and thus is not provided. To gain a better understanding of the flow field on a longwall face, full scale three dimensional models have been developed. For the first time, the models were constructed as close as possible to the actual longwall face geometry by incorporating the key features of the longwall equipment in the models. As the dispersion and migration of hazardous gas and dust are highly dependent on the airflow patterns, it is of great significance to obtain a more realistic airflow field by incorporating all the key features of major longwall equipment in the models.

Based on data collected from an Australian longwall face, a three dimensional longwall CFD model was constructed. Figure 4.1 shows a general layout of the longwall model. The model was developed by incorporating all the main equipment in the face, including 132 longwall chocks, shearer, AFC, Beam Stage Loader (BSL) and breaker/feeder of the belt conveyor. The longwall model represents a working scenario when the shearer is cutting from the MG to TG and the MG drum is 10 m away from the MG rib. In addition, the longwall operation is conducted in a seam with a thickness of 3.5 m, and the 200 m long face is supported by 132 chocks. As the focus of this study is on ventilation characteristics along the face, only 50 m long of MG and TG are included in the model. The last cut through on the MG side is located 45 m outbye the face whilst the cut through on the TG side is 10 m away from the face line on the goaf side. The cut throughs, MG and TG share the same width of 5 m. It is noted that the location of the TG cut through may not be consistent with the actual panel layout, here it is used for the purpose of investigating the gas flow patterns under different ventilation systems in the following chapter. The
position of the MG cut through is in accordance with the position where an actual ventilation survey was conducted for base model validation.

Compared with previously reported longwall models (Srinivasa, 1993; Zheng and Tien, 2009; Ren, et al., 2012), the CFD model developed in this study is further improved in the following aspects:

- The hydraulic legs for all longwall chocks are included in the model as extracted solid parts, rather than a volume of ‘porous’ block in previous longwall models (Ren, et al., 2012), which are considered important to produce more realistic modelling results of airflow patterns along the longwall face;
- The majority of equipment which may affect the airflow patterns along the longwall face have been incorporated in the model, these include the belt conveyor and BSL in the MG, the chocks, AFC and the shearer along the face;
- Considering the impact of air exchange between the face and goaf on face ventilation and subsequently the hazardous gas and dust flow patterns, a 10 m long goaf immediately behind the chocks is included in the model;
- A cut through is added to the TG side of the model (denoted as TG ct) to investigate the impact of different face ventilation systems on the gas flow behaviour at the longwall face, especially at the intersection of the longwall face and the TG;
- The shearer’s drums are constructed to be close to the general configuration of actual drums, and sprays on the drums behind the cutting picks are added to the model, thus, the influence of these sprays on the gas and dust flow patterns around the shearer can be investigated;
- By attaching sprays to the canopy of longwall chocks, BSL transfer point and on the shearer body, the dust suppression effect under different operating conditions of the sprays can be assessed, providing guidance to the field implementation of sprays for better dust control.
To thoroughly understand the airflow patterns along the longwall face, two more models have been developed to investigate the flow characteristics under different longwall operating scenarios, which are achieved by positioning the shearer to the middle section and TG side (refer to the potential shearer positions illustrated in Figure 4.1a) of the longwall face respectively. In addition, as the airflow pattern around the shearer is significantly affected by the cutting sequence, three more models are constructed to represent scenarios when the shearer is cutting from TG towards MG. For the three models indicating the TG to MG pass, the shearer in each model is located at the same position as the corresponding MG-TG pass models. Figure 4.2 shows the scenario of a longwall model when the shearer is cutting from TG to MG and is close to the MG. The six models representing six different working scenarios of the longwall face are given a distinctive name, as shown in Table 4.1.

(a) Plan view of the longwall model - shearer close to MG

(b) 3D view of the longwall model - shearer close to MG
Upon defining the physical geometry of the longwall face, the model was meshed using the unstructured tetrahedron approach to accommodate its complex geometry. The same mesh controls on the boundaries and the domain were applied to all the six models, leading to an approximately equal size of 2.2 million control volumes (or computational cells) for each model. Figure 4.3 depicts the computational grid.
adopted for the model where relatively dense mesh has been employed around the shearer, especially in the vicinity of the drums and face ribs close to the drums.

Table 4.1 Short name for the six longwall models

<table>
<thead>
<tr>
<th>Model Name</th>
<th>Operating scenario</th>
</tr>
</thead>
<tbody>
<tr>
<td>MG-TG Case 1</td>
<td>Shearer is cutting from MG to TG and is located close to the MG</td>
</tr>
<tr>
<td>MG-TG Case 2</td>
<td>Shearer is cutting from MG to TG and is located in the middle</td>
</tr>
<tr>
<td>MG-TG Case 3</td>
<td>Shearer is cutting from MG to TG and is located close to the TG</td>
</tr>
<tr>
<td>TG-MG Case 1</td>
<td>Shearer is cutting from TG to MG and is located close to the MG</td>
</tr>
<tr>
<td>TG-MG Case 2</td>
<td>Shearer is cutting from TG to MG and is located in the middle</td>
</tr>
<tr>
<td>TG-MG Case 3</td>
<td>Shearer is cutting from TG to MG and is located close to the TG</td>
</tr>
</tbody>
</table>

Figure 4.3 Computational grids adopted for the TG-MG Case 1 longwall model

(a) Model grids adopted at the MG corner

(b) Model grids adopted around the shearer
The boundary types are defined during the meshing process so that different boundary conditions can be conveniently applied in the CFD modelling process. As shown in Figure 4.1, fresh air is provided to the face through the MG and the MG cut through, and these two inlets are treated as velocity inlet. As the main outlet of the model, the TG is treated as pressure outlet. While for the TG cut through which may be used for gas discharge, velocity inlet is used to allow a certain amount of gas mixture going out when necessary. Velocity inlet is also adopted for the sprays boundary condition, and zero velocity is used when the sprays are not working. For the moving parts of the longwall equipment, the shearer drums are treated as a moving wall with a rotational speed of 35 rpm to investigate its local influence on airflow patterns in the vicinity of the shearer, whilst the surfaces of the AFC and belt conveyor are also set as moving wall but with a translational speed of 1.78 m/s and 4m/s respectively. For the other boundaries of the model, such as the coal ribs, floor, roof and chocks, standard wall conditions are employed. It is worth noting that in this calculation the flow medium (air) is assumed to be incompressible and the temperature field is constant, thus the inlet ventilation velocity can be calculated for both MG and the MG cut.

In addition, the extension of longwall face model to include a short goaf constitutes another challenge besides the construction of longwall models with such a complex geometry. As the goaf is generally filled with broken rocks as the roof strata collapse, airflow leakage through the goaf travels much slower than that in the face, and this is achieved by adding a momentum sink to the momentum equation. However, the momentum sink across the goaf is not constant but varies with positions in the goaf, depending on the degree of compaction. Due to the effect of the old support system for the collapsed MG and TG in the goaf, the roof above the old MG and TG are not collapsed/broken as well as the roof in the centre of the goaf, thus, the degree of compaction in the centre of the goaf is higher than that along the perimeter of the goaf, which determines that the momentum sink along the perimeter of the goaf is relatively less than that in other areas of the goaf. Meanwhile, the degree of goaf compaction immediately behind the chocks is lower than that in the deep goaf. Following this principle, a user defined function (UDF) subroutine was written to
represent the momentum sink in the goaf which varies with coordinates of the computational grids.

4.3 Modelling fluid flow in underground ventilation system

4.3.1 Flow regimes in the underground ventilation system

Before running the calculation, it is important to identify the flow regimes of the longwall ventilation system, i.e., laminar flow or turbulent flow. In fluid mechanics, the Reynolds number $R_e$ is widely used to identify whether the flow is laminar or turbulent. It is defined as the ratio of inertial forces to viscous forces, and has the following form:

$$R_e = \frac{\rho V L}{\mu} = \frac{V L}{v}$$  \hspace{1cm} (4.1)

where, $R_e$ is the Reynolds number, dimensionless;
- $V$ is the mean fluid velocity, m/s;
- $L$ is a characteristic length, m;
- $\mu$ is the dynamic viscosity of the fluid, Pa•s;
- $v$ is the kinetic viscosity ($v = \mu/\rho$), m²/s;
- $\rho$ is the density of the fluid, kg/m³.

Depending on the value of the $R_e$, the flow regime can be determined. For example, if $R_e<2000$, the flow is considered to be laminar and if $R_e>2300$, the flow is turbulent.

For the airflow in the underground ventilation system, the Reynolds number can be calculated using Eq. 4.1. It is worth noting that the hydraulic diameter is used as the characteristic length in the calculation. For the rectangular roadway which is completely filled with flow medium, its hydraulic diameter is defined as

$$D_h = \frac{4HW}{2(H+W)} = \frac{2HW}{H+W}$$  \hspace{1cm} (4.2)
where, $D_H$ is the hydraulic diameter of the roadway, m;

$H$ is the height of the roadway, m;

$W$ is the width of the roadway, m.

Therefore, for the case of this study where the roadway is 3.5 m high and 5 m wide, the hydraulic diameter can be calculated according to Eq. 4.2 and is equal to 4.12 m.

At 15°C under the standard atmospheric pressure, the kinetic viscosity of air is $1.47 \times 10^{-5}$ m$^2$/s. Thus, assuming the mean air velocity along the face is between 2 m/s and 3 m/s, the corresponding Reynolds number can be determined by Eq. 4.1 and it is between $5.61 \times 10^5$ and $8.4 \times 10^5$, indicating the airflow along the face is absolutely/fully turbulent; On the other hand, the minimum velocity (denoted as $V_{min}$) required for the flow to be in the turbulent regime can also be determined using the deformation of Eq. 4.1, which can be expressed and calculated as:

$$V_{min} = \frac{Ry}{L} = \frac{Ry}{D_H} = \frac{2300 \times 1.47 \times 10^{-5}}{4.12} = 0.0082 \text{ m/s}$$

(4.3)

It is known that the minimum flow velocity required in the underground ventilation system is above 0.15 m/s. Consequently, it can be concluded that the flow in the entire ventilation system is categorised into the turbulent flow regime, except in the deep goaf where the flow velocity may be lower than 0.0082 m/s.

4.3.2 Turbulence model selection

Knowing the fact that the flow is turbulent in the roadway, it is imperative to understand the background of different turbulence models provided in ANSYS Fluent, based on which a reliable flow field is obtained. Three computational approaches are generally used to solve the Navier-Stokes equations, i.e., Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and the Reynolds-Averaged Navier-Stokes (RANS) models. However, due to the extremely high requirements of the DNS and LES approaches, they are rarely adopted in the actual
large scale flow simulations, especially the DNS approach which demands very fine
grid and very small time step and thus is restricted to supercomputer applications for
lower Reynolds-number flows. As a result, in this study, the most widely used
approach – the RANS models are adopted which constitute the focus of the following
section. It is noted that the following partial differential equations are heavily
referred to the ANSYS Fluent manual (ANSYS, 2010).

In Reynolds averaging, the variables of the instantaneous field are decomposed into
the mean and the fluctuating components, such as:

\[ u_i = \bar{u}_i + u'_i \]  \hspace{1cm} (4.4)

where \( \bar{u}_i \) and \( u'_i \) are the mean and fluctuating velocity components.

Therefore, the RANS equations can be obtained by taking a time average of the
Navier-Stokes equations:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j} (-\rho u'_i u'_j) \]
\hspace{1cm} (4.6)

As can be seen from Eq. 4.6, a new variable, the Reynolds stress \(-\rho u'_i u'_j\) is
introduced to the equations and it must be solved concerning the closure of the
equations. Two approaches are adopted to calculate the Reynolds stress, i.e., the
Boussinesq hypothesis and the Reynolds Stress Models (RSM).

In the Boussinesq hypothesis, the Reynolds stresses are related to the mean velocity
gradients:
Based on the Boussinesq hypothesis, several turbulence models, such as the Spalart-Allmaras model, the $k-\varepsilon$ model and the $k-\omega$ model, are provided in the Fluent, which vary from each other in determining the turbulent viscosity $\mu_t$. And in the Spalart-Allmaras model, only one additional equation is solved to obtain the turbulent viscosity $\mu_t$. In the $k-\varepsilon$ model and the $k-\omega$ model, two additional equations are employed to calculate $\mu_t$.

However, in the RSM, the transport equations for each of the terms in the Reynolds stress tensor are solved, indicating five and seven additional equations are required to be solved for 2D and 3D flows respectively.

Comparisons between the two approaches indicate that the RSM is superior to the Boussinesq hypothesis based models when the effect of anisotropy of turbulence is dominant on the mean flow, such as highly swirling flows and stress-driven secondary flow (ANSYS, 2010).

Considering the flow characteristics in the underground ventilation system, the Boussinesq hypothesis based $k-\varepsilon$ model is adopted in the study. It is noted that the $k-\varepsilon$ model is subdivided into the Standard $k-\varepsilon$ model, the RNG $k-\varepsilon$ model and the Realizable $k-\varepsilon$ model. Comparison calculations were conducted using the three sub models and no significant variation was observed among the corresponding results, thus, the Standard $k-\varepsilon$ model is finally used.

4.3.3 Transport equations for the Standard $k-\varepsilon$ model

In the standard k-ε model, the turbulent viscosity, $\mu_t$, is determined by the following equation:
\[ \mu_i = \rho C_{\mu} \frac{k^2}{\varepsilon} \]  

(4.8)

where \( C_{\mu} \) is a constant, \( k \) is the turbulence kinetic energy and \( \varepsilon \) is the turbulence dissipation rate.

Meanwhile, the transport equations used for the determination of \( k \) and \( \varepsilon \) can be written as (ANSYS, 2010):

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[ \left( \frac{\mu + \frac{\mu}{\sigma_k}}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{4.9}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[ \left( \frac{\mu + \frac{\mu}{\sigma_\varepsilon}}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1_\varepsilon} \frac{\varepsilon}{K} (G_k + C_{3_\varepsilon} G_b) - C_{2_\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \tag{4.10}
\]

where, \( G_k \) represents the production of turbulence kinetic energy due to the mean velocity gradient; \( G_b \) is the generation of turbulence kinetic energy due to buoyancy; \( Y_M \) represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate; \( C_{1_\varepsilon}, C_{2_\varepsilon} \) and \( C_{3_\varepsilon} \) are constants. \( S_k \) and \( S_\varepsilon \) are user-defined source terms.

From the exact equation for the transport of \( k \), the term \( G_k \) may be defined as (ANSYS, 2010):

\[ G_k = -\rho u_i' u_j' \frac{\partial u_j}{\partial x_i} \]  

(4.11)

The generation of turbulence kinetic energy due to buoyancy can be determined by (ANSYS, 2010):

\[ G_b = \beta g_i \frac{\mu_i}{Pr_i} \frac{\partial T}{\partial x_i} \]  

(4.12)
where, $\beta$ is the thermal expansion coefficient, and is defined as $\beta = -\frac{1}{\rho} \left( \frac{\partial \rho}{\partial T} \right)_p$;

$Pr_t$ is the turbulent Prandtl number for energy, and a default value of 0.85 is adopted for the standard $k-\varepsilon$ model;

g$_i$ is the gravitational components in the $i$th direction.

### 4.4 Base model validation and results

Due to the availability of limited velocity profiles obtained from field monitoring, only the MG-TG Case 1 model can be used for validation. Therefore, only the validation of the MG-TG Case 1 is described here, and the result is used to assess the feasibility and reliability of investigating the longwall face flow patterns using similar CFD models.

The field monitoring was carried out in the conditions illustrated in Figure 4.1a: the shearer is cutting from MG to TG and the MG drum is 10 m away from the MG rib, the longwall face is ventilated with 45 m$^3$/s air, of which 33 m$^3$/s from the last cut through (MG ct), but there is no TG ct for field condition. So the TG ct is switched to a solid zone in the calculation not allowing airflow passing through it. Under this condition, the MG-TG Case 1 model is referred to as the base model in this section, and the following parametric studies conducted in section 4.5 are all relative to the base model in terms of changing ventilation rate or ventilation system or some other conditions.

#### 4.4.1 Base model validation

During the field ventilation survey, the distribution of airflow velocity at five cross-sections along the MG and face (mostly around the shearer) was measured and subsequently used to validate the base model. Figure 4.4 shows a comparison of model predicted results and field measured velocity at the five cross-sections.
(a) Velocity distribution at 5 m outbye the face (in MG)

(b) Velocity distribution at 1 m from MG

(c) Velocity distribution at 2 m before the MG drum

(d) Velocity distribution in the middle of the shearer
It can be seen from Figure 4.4a, due to the space constraints caused by BSL/crusher and associated equipment, the airflow travels a bit faster along the walkway in the MG before entering the face, this may lead to serious air leakage into the goaf if no goaf curtain is used in the gap between the chocks and the MG rib. As the ventilation enters the longwall face with relatively larger cross areas, it then travels at a slightly lower velocity, with the maximum value around 3.5 m/s along the face.

It is worth noting that the measured velocity profiles on different cross section along the face only indicate the velocity magnitude along the face, which is also the major component of airflow velocity along the face. In addition, the airflow patterns at the MG corner are very complex due to the turbulence nature of fluid flow and the occurrence of flow separation phenomenon, as shown in Figure 4.4b. Thus, rather than using the resultant velocity magnitude, the velocity component in the face length direction predicted by the model was used for base model validation. That is also the reason why both negative and positive values appear in the velocity scales.

In this case, the negative value indicates the air flows towards the TG while the positive value means the air flows towards the MG. It is noticed that the impact of the AFC on airflow field is only confined to a limited area above it, whereas the effect of flow separation is significant at the MG corner. Regarding the flow patterns at the MG corner and the occurrence of the flow separation phenomenon, more details will be provided in the following section.
Generally, the model predicted velocity profiles agree quite well with the monitored ventilation velocity data, indicating a proper setup of the longwall model has been achieved and the model predicted results are reliable for the investigation of flow patterns along the longwall face. Meanwhile, it is also observed from both model predicted and the monitored velocity profiles that airflow velocities are not evenly distributed across these sections. As the flow enters the face, the velocity is relatively higher in the walkway, then the high velocity area gradually migrates towards the upper section of face side until it passes the shearer body; however, within 2 m behind the TG drum, the higher velocity area appears above the spill plate.

4.4.2 Base model results

The validation of the base model revealed some information about the flow patterns around the MG corner and the shearer, however, this information is not sufficient to describe the flow characteristics at these positions, let alone the flow characteristics along the entire face. Thus, a detailed analysis of the airflow field along the longwall face is provided in the following section.

4.4.2.1 Flow separation along the longwall face

Before investigating the airflow patterns along the longwall face, it is important to firstly obtain a brief idea of the flow separation phenomenon which causes additional energy losses to the ventilation system and accumulation of hazardous gas and dust. Flow separation commonly occurs in the face ventilation system; however, its potential risk to the gas management of face has not attracted much attention from ventilation engineers.

As can be seen from Figure 4.5, when fluid flow passes the solid boundary of an object, a very thin layer will be formed in the immediate neighbourhood of the solid boundary, in which the velocity of the fluid increases from zero at the wall to its fully developed value away from the wall. This thin layer is termed as a boundary layer in fluid mechanics (Schlichting and Gersten, 2000). There is no doubt that the boundary
layer is defined relative to the viscous fluid flow and it commonly exists in both external and internal flows.

Consider the situation when an external flow over a flat object encounters a positive pressure gradient in the flow direction (which is also called an adverse pressure gradient), the flow will be decelerated, and in the boundary layer it will be decelerated much more than the mean flow. Once the velocity of the boundary layer fails to overcome the adverse pressure gradient, the velocity near the wall becomes zero or even negative, leading to the separation of the boundary layer from the wall and backflow appears in the downstream. From the perspective of fluid mechanics, this phenomenon is defined as flow separation. A steady state of flow separation process under a given pressure distribution is illustrated in Figure 4.6 (Herbert, et al., 2009). It is noted that in Figure 4.6, the separation starts at point A (termed as separation point) where the velocity gradient perpendicular to the wall vanishes, i.e., 

\[ \frac{\partial u}{\partial y}_{\text{wall}} = 0, \]

where \( u \) is the velocity and \( y \) indicates the direction normal to the wall (refer to Figure 4.5). As a result, the wall shear stress (\( \tau \)) at the separation point becomes zero too (\( \tau = \mu \frac{\partial u}{\partial y}_{\text{wall}} = 0 \)).

In most cases, the flow separation is undesirable as it cost energy losses. For example, the majority of vehicles are designed to have a streamlined appearance, especially
racing cars, aiming at reducing the pressure drag induced by the flow separation around them. The same design principle also applies to airfoils and submarines.

Figure 4.6 The flow separation process (Herbert, et al., 2009)

Though flow separation is undesirable, it is very common in the fluid related flows, and there is no exception for the underground ventilation system, in which it may cause safety issues, i.e., gas and dust accumulation. Owing to the complex spatial relations among the roadways, flow separation generally occurs at the intersections of roadways connected with various angles. Regardless the occurrence of separation in the entire ventilation system, more attention has been paid to the flow separation on the longwall face, especially at the MG and TG corners, where the face is connected to the MG and TG at an angle of 90 degrees.

Take the MG corner as an example, as the airflow passes the MG, boundary layers will be formed near the ribs, floor and roof, for simplification, the flow is considered in two dimensions. Then the airflow around the MG corner can be illustrated schematically as Figure 4.7. As the flow approaches point B (the intersection of face rib and MG rib) and is about to enter the longwall face, the boundary layer near the MG rib AB will experience an adverse pressure gradient during the turning process from the MG to face. Since the kinetic energy in the boundary layer is so small that it cannot surmount the adverse pressure gradient, the flow is thus separated in the downstream towards face, which implies the mean airflow flows towards the face while in the area close to the face rib BC the flow reverses to the upstream towards
point B. Therefore, a recirculation zone is generated in the vicinity of face rib BC. As the air flows further downstream, the thickness of boundary layer gradually diminishes to a stable value and the backflow disappears, then the flow along the face returns to normal channel flow except in the vicinity of the longwall shearer or the cutting ribs where smaller scales of separation may occur compared with the separation at the inner MG corner. Figure 4.8 depicts the flow patterns around the corner which clearly shows the occurrence of flow separation at the inner MG corner. It is noticed from Figure 4.8 that no obvious separation is predicted at the outer MG corner owing to the air leakage to the goaf. Meanwhile, it is also observed that the mean airflow is accelerated around the MG corner. This can be attributed to the flow separation at the inner corner which introduces backflow towards upstream. In order to satisfy the incompressible continuity requirement, the mean flow away from the backflow area must be accelerated accordingly.

As can be seen from Figure 4.9, at the face to TG corner, similar flow patterns to the MG corner is predicted, and the flow separation mechanism also applies to the flow at the TG corner. What makes a difference is that the separation occurs in the vicinity of inner TG rib rather than the face rib. Meanwhile, it is noted that the acceleration of airflow at the outer TG corner is more significant than that at the MG corner (face entrance), making it the highest velocity area in the entire modelled longwall ventilation system. This is due to the variation of air exchange patterns between the face and goaf. At the MG corner, a certain amount of airflow is able to penetrate through the permeable goaf; however, this part of airflow will return back to the face further down along the face until it reaches the TG corner which is the last entrance for it to rejoin the main return flow. Thus, the flow rate at the TG corner is larger than that at the MG corner (face entrance); with similar cross sections, the flow is much more accelerated near the outer TG rib.

As mentioned before, the occurrence of flow separation in the ventilation system is not desirable either, not only because it causes energy loss but also increases the likelihood of hazard gas and dust accumulation in the recirculation zone. The specific impact of flow separation on the gas accumulation at the two corners will be investigated in the following chapter.
Figure 4.7 Flow schematic at the MG corner

Figure 4.8 Flow patterns at the MG corner indicating airflow separation - 2m above floor level
4.4.2.2 General airflow patterns

Figure 4.10 shows the overall airflow velocity contour along the entire face at different levels above the floor. As expected, the airflow velocity is not evenly distributed along the face. Areas along the MG, around the MG and TG corner and within 10 m downwind of the shearer are typically the major high velocity areas. In the MG, the velocity is slightly higher due to the existence of equipment in the roadway. While at the MG and TG corner where the separation occurs, the velocity is slightly higher at the outer corners to maintain the continuity of the incompressible fluid. It is observed from these modelling results that the majority of airflow, which has penetrated to the goaf at the MG corner, returns to the main flow again just...
behind the shearer. Therefore, the flow rate in the downstream of the shearer recovers to approximately the same flow rate at the MG regardless of the quantity of flow lost in the goaf. Hence, the increased flow rate together with the reduced cross section contributes to the relatively higher velocity the downstream of the shearer, and this effect is more significant at higher levels.

It is also observed that the velocity distribution in the face retreating direction can be categorised into three sections. Within the goaf, the overall velocity is lower than 0.8 m/s, except the area behind the MG corner where it can reach as high as 1.7 m/s. In the walkway under the longwall chocks, an average velocity of 2.5 m/s is observed downstream of the shearer whilst upstream of the shearer the velocity is slightly lower at around 2 m/s owing to the leakage to the goaf. Above the AFC, the ventilation flow travels a bit faster at about 3 m/s.

(a) Velocity contour on the surface 1.5 m above floor (near the top of the shearer)
As the only exit, the airflow pattern in the TG is of great significance in terms of discharging the hazardous gas and dust associated with the longwall coal extraction.
As is shown in Figure 4.10, the influence of separation at the inner TG corner is significant in the first 10 to 15 m even though the main backflow occurs within 4 to 5 m. The backflow and stagnant air in the downstream will increase the risk of gas accumulation. It is also inferred that the majority of gas trapped in the recirculation zone will be the gas liberated from the longwall face rather than the goaf. For the gas emitted from the goaf, it is likely to accumulate in the sluggish zone under the TG chocks and the gap between the last chock and the TG rib. Thus, effective control measures are recommended to induce the gas at the outer TG corner to be returned to the airflow, which would dilute the gas instantaneously by the accelerated flow near the outer TG rib. It is important to identify the gas sources that lead to different levels of gas accumulation at different positions, and this will help in developing effective gas control strategies under different mining conditions. It is worth noting that the above inference is obtained from the velocity contour at the TG corner; however, more evidence is needed to verify whether the inference is right or not. More detailed information about gas accumulation will be provided in the following chapter.

To obtain a better understanding of the air exchange pattern between the longwall face and goaf, the fluid streamlines from the MG are illustrated in Figure 4.11. It can be seen that air leakage mainly occurs at the MG corner where the goaf is not sufficiently compacted. As the goaf compaction level increases in the deep or middle of goaf, little air is able to penetrate it. The major proportion of leaked air flows back to the face at around 50 m down the face (directly behind the shearer position in this case). In the following 100 m along the face, no significant air exchange is observed except some small fluctuations around the back legs of longwall chocks; Whereas further downstream along the face, a small quantity of fluid starts to ingress into the goaf owing to the unconsolidated goaf condition near the TG. As the flow approaches the TG, separation occurs at the inner TG corner, whilst at the outer TG corner a triangle zone (refer to the red triangle in Figure 4.11) occupied by approximately stagnant air is formed, both of which are undesirable from the perspective of gas management at longwall face.
It is noted that in this case, the airflow is provided to the face through both the MG belt roadway and the MG ct, the MG ct provides the majority of flow rate. As a result of the liberation of gas during the transport process of coal in the MG, the intake airflow will contain a certain amount of gas. Figure 4.12 depicts the flow streamlines from the MG belt roadway. It can be seen that the airflow is pushed to the inner MG rib by the flow from the MG ct in the MG. As the flow enters the face, part of it is involved in the recirculation zone resulting from airflow separation, which may lead to a build-up of methane gas at higher concentrations within that zone; whilst the majority of the flow is constrained to the zone close to the face. This airflow pattern could potentially lead to gas management problem; however it is favourable for respirable dust control because dust particles contained in the airflow from the MG belt roadway will be diverted away from the walkway.

Figure 4.11 Airflow streamlines (form both the MG and the MG ct)
4.4.2.3 Airflow patterns around the shearer

At the longwall face, as coal is extracted by the shearer, the majority of hazardous gas and dust is generated in the coal extraction process, thus it is important to investigate the airflow patterns around the shearer. As mentioned before, a longwall shearer has been incorporated in the model, aiming at capturing as much details of the flow as the field situation. It is noted that the velocity scale adopted in the following sections is the same as the one used in Figure 4.10 unless specified otherwise.

Figure 4.13 illustrates the velocity contour around the shearer at different levels above the floor. As expected, the air flow velocity is not evenly distributed around the shearer as the air passes the shearer body. A small separation zone is observed ahead of the TG drum; however, this zone gradually diminishes as the elevation increases, indicating the airflow separation is mainly induced by the shearer body at the lower position. The ranging arms also have a significant effect on the flow field, leading to stagnant airflow in the downstream and accelerated airflow in the walkway. The rotation of shearer drums provides an additional motion to the flow, stimulating the airflow movement in the vicinity of the drums where the gas emission...
rate is supposed to be the highest along the face. Above the shearer body, the velocity gradually increases as the distance above it increases, two factors are thought to be the cause of this characteristic; As the airflow travels past the shearer body, the effective cross section is reduced due to the occupation of shearer body, whilst the leaked air in the goaf flows back to the face through gaps between the chocks, pushing the airflow towards the face side and upwards.

(a) Velocity contour near the top of the shearer - 1.5 m above floor

(b) Velocity contour at 0.5 m above the shearer - 2 m above floor

(c) Velocity contour at 1 m above the shearer - 2.5 m above floor
Figure 4.13 Airflow velocity contour around the shearer

Figure 4.14 provides a 3D view of the velocity vectors around the shearer. It can be seen that at lower elevations, the ventilation is able to flow across the gaps between longwall supports and re-enter the face to join the main air stream. Owing to the special configurations and the location of the shearer body, the flow direction varies as the flow passes the shearer, indicating an “S” shape airflow pattern. It is observed that the intake air flows towards the face rib at the MG drum position and above it due to the influence of separation at the MG corner. As the airflow approaches the TG drum, the velocity becomes smaller near the drum, and the flow is deflected to flow faster towards the walkway, after which it flows back to the old face rib and starts to travel straight along the face.

Figure 4.15 shows the distribution of flow vectors around the shearer drums when the drums are rotating at a speed of 35 rpm. The impact of drum rotation on regional airflow patterns can be observed. It is also noticed that the rotating direction of the drums affects the velocity distribution to some extent. The velocity is slightly reduced in the vicinity of the MG drum (the vector colour varies from blue to light blue) whereas it is increased in the vicinity of the TG drum (the vector colour varies from green to light yellow), as well as the velocity distribution parallel to the drums. Therefore, it can be concluded that airflow is adversely affected by the MG drum rotation which exerts a negative motion to the airflow; while the flow is substantially accelerated by the TG drum rotation which exerts a positive motion to the airflow.
(a) Near the top of the shearer  
(b) 0.5 m above the shearer  
(c) 1 m above the shearer  
(d) 1.5 m above the shearer  

Figure 4.14 3D view of airflow velocity vectors around the shearer
4.4.2.4 Velocity distribution cross the face

Figure 4.16 shows the velocity variations from the face rib towards the goaf at four different locations along the face, i.e., 1 m face inbye on the MG side, the middle of shearer body and face, and 1 m face inbye on the TG side. The 2 m level is chosen to stand for the workforces’ breathing level. It can be seen that the overall velocity varies significantly from the face to the goaf, and even in the face area, the velocity is not evenly distributed, especially just after the flow enters the longwall face where a large volume of airflow intends to penetrate into the goaf. Generally, the flow regimes can be categorised into three sections which are in accordance with the geometry of the face, namely, space along the face, space between the chocks’ legs
and the immediate goaf behind the chocks, as indicated in Figure 4.16. It can be observed that the overall velocity in the goaf is the lowest in the three sections, with the maximum velocity of 0.5 m/s occurring at the MG end, indicating a significant air exchange between face and goaf, whilst the velocity is lower than 0.25 m/s in the goaf as the flow travels to the TG, reflecting a good roof caving and compaction in the goaf.

Figure 4.16 Velocity distribution at different locations across the face - 2 m above the floor

4.5 Parametric studies

The base model reveals a working scenario of the longwall extraction process when the shearer is cutting from MG to TG and located close to the MG; however, as the shearer is cutting further downwards, the corresponding airflow patterns may vary accordingly. Thus, the following part describes the airflow patterns along the longwall face considering more working scenarios in terms of shearer position and cutting sequence.
4.5.1 Impact of shearer position

Figure 4.17 and Figure 4.18 respectively show the velocity contour at 2 m above the floor for the MG-TG Case 2 and Case 3 (refer to Table 4.1). It can be seen that the general airflow pattern is similar to that of the base model. The location of the shearer only has a marginal effect on the general airflow patterns. In both models, there has been significant air leakage into the goaf at the MG corner as the base model. Due to the leakage, the ventilation at the first 20 to 30 m of face is slightly weaker than other parts of the face, characterised by lower velocity compared with the flow further downstream. With the gradual flow back of the leaked air, the velocity gradually increases to about 3.0 m/s, and then becomes stable as it flows along the face until it reaches the shearer body where the velocity starts to fluctuate again. A similar velocity contour distribution pattern to the base model is predicted around the shearer in both models.

Figure 4.17 Velocity contour at 2 m above floor for MG-TG Case 2

Figure 4.18 Velocity contour at 2 m above floor for MG-TG Case 3
The velocity vectors around the shearer for both models are illustrated in Figure 4.19 and Figure 4.20 respectively. As can be seen, the general flow patterns are slightly different with that of the base model. Along the longwall face, the flow shows a “C” shape pattern rather than the “S” shape as in the base model. This is because the impact of airflow separation vanishes as the air flows further to the face, and when the shearer reaches the middle or close to the TG, the air stream has been flowing straight along the face upstream of the shearer.

![Figure 4.19 3D view of velocity vectors around the shearer for MG-TG Case 2](image)

(a) Near the top of the shearer  (b) 1 m above the shearer

It is also noted that the flow immediately behind the chocks is different for all three models. In the base model, the ventilation tries to return back to the face across the whole length of the shearer body (refer to Figure 4.14a) and rejoin the main flow. When the shearer is cutting in the middle of the longwall face, the main flow starts to penetrate into the goaf because of the chocks advance and the obstruction of the shearer body, resulting in increased velocity between the chock legs. After the leaked airflow travels past the shearer body, it flows back towards the face. As the shearer is cutting close to the TG, the flow along face has already started penetrating to the goaf upstream of the shearer body, and the penetration will be accelerated by the chocks advancement as that in the MG-TG Case 2. In addition, in MG-TG Case 3, the
ventilation leaked to the goaf flows straight towards the TG side instead of flowing back to the face as the case of MG-TG Case 2. Consequently, the flow downstream of the shearer is less accelerated in MG-TG Case 3. This kind of flow pattern is also well interpreted on the velocity contour in Figure 4.17 and Figure 4.18.

![3D view of velocity vectors around the shearer for MG-TG Case 3](image)

(a) Near the top of the shearer  (b) 1 m above the shearer

Figure 4.20 3D view of velocity vectors around the shearer for MG-TG Case 3

4.5.2 Impact of cutting sequence

It is known from section 4.5.1 that the shearer position has a local effect on the flow patterns around the shearer, whereas it does not cause much disturbance to the overall flow patterns. The cutting sequence will also substantially affect the airflow patterns in the vicinity of the shearer.

Figure 4.21 illustrates the velocity contour at different levels above the floor for TG-MG Case 1. As stated before, there is no significant difference in the overall flow patterns when compared with the base model results, except in the region around the shearer body. It is observed that the flow parallel to the shearer body is accelerated but not as much as it is in the base model, especially in the area next to the ranging
arm of the leading drum. In the base model, the leading drum is the TG drum, whilst in the TG-MG Case 1 model, the MG drum becomes the leading drum. Though the leading drum has changed for the two models, the rotating direction of each drum does not change. As noted in section 4.2.3, an adverse flow motion to the airflow direction is generated during the rotation of the MG drum, while a positive flow motion is formed during the rotation of the TG drum. In both cases, the impact of the leading drum is dominant and outweighs the other drum. Consequently, the flow is less accelerated as the flow passes the shearer body and even downstream. It is worth noting that in the TG-MG pass, the advance of chocks behind the TG drum causes the air to flow through the space between the chocks legs, increasing the air loss to the goaf.

(a) Velocity contour on the surface 1.5 m above floor (near the top of the shearer)

(b) Velocity contour on the surface 2 m above floor
Figure 4.21 Velocity contour at different levels above floor for TG-MG Case 1

Figure 4.22 depicts the velocity vectors around the shearer, from which it can be seen that there is no significant flow deflection at higher elevation even near the ranging arm; however, at lower levels, small fluctuations of the flow direction are shown due to the chocks advancement. The same as base model, affected by the air leakage in the goaf, the flow behind the chocks are flowing back to the face.
Figure 4.22 3D view of velocity vectors around the shearer for TG-MG Case 1

Figure 4.23 and Figure 4.24 respectively show the velocity contour distribution for TG-MG Case 2 (shearer in the middle of longwall face) and Case 3 (shearer close to TG). It can be seen that the overall flow pattern away from the shearer is similar to the other models. The velocity is slightly higher in areas close to the face rib than it in the walkway. However, as the flow travels past the shearer, due to the obstruction of the leading drum and its ranging arm, the high velocity airflow near the face rib is deflected to the walkway side, and the area behind the leading drum and its ranging arm is decelerated, generating a large area with low velocity above the shearer body and close to the fresh rib. In the MG-TG pass where the flow is also deflected by the ranging arm of the leading drum (TG drum), the deflected flow can attach to the old face rib within a short distance downstream; while in the TG-MG pass, the deflected flow needs to travel a longer distance to be able to attach to the fresh rib which is one web depth further in the coal seam than the old face rib. Once the high velocity flow attaches to the face rib again, the disturbance of the shearer to the flow field comes to an end, and the flow gradually recovers to the state before it passes the shearer. For TG-MG Case 3, it is observed that the high velocity flow near the face rib cannot
recover until it reaches the TG owing to the increased air leakage to the goaf as the flow approaches the TG.

![Figure 4.23 Velocity contour on the surface 2 m above floor for TG-MG Case 2](image)

![Figure 4.24 Velocity contour on the surface 2 m above floor for TG-MG Case 3](image)

The velocity vectors around the shearer for TG-MG Case 2 and Case 3 are illustrated in Figure 4.25 and Figure 4.26 in 3D view. Compared with TG-MG Case 1, there is no significant difference in the airflow patterns along the face, and they vary from each other in the flow behind chocks as the three models in the MG-TG pass. However, there is a significant difference between the two cutting sequence models. The shearer moves against the flow as it cuts from the TG to MG, thus the advancement of chocks intercepts the flow in the walkway, causing more air leakage to the goaf. As can be seen from Figure 4.25 and Figure 4.26, though the flow has passed the shearer, the vectors still indicate a strong trend of migration to the goaf near the last advanced chocks.
Figure 4.25 3D view of velocity vectors around the shearer for TG-MG Case 2

Figure 4.26 3D view of velocity vectors around the shearer for TG-MG Case 3

4.5.3 Impact of curtain on MG side

Model results from all the six models described above indicate significant air leakage to the goaf occurs at the MG corner. The leakage is undesirable in the ventilation
system and it may cause severe safety issues if not managed properly, i.e. loss of air ventilation on the face and spontaneous combustion of coal in the goaf.

In the base model, a 7 m long curtain is added behind the front legs of the chocks. It extends from the outer MG rib to the fourth chock in the face. In the modelling process, two approaches can generally be used to simulate the operating scenario with a MG goaf curtain. The first approach involves incorporating a solid wall at the MG corner which requires modification to the physical model and re-meshing of the new model; whilst the second approach uses a subroutine to mimic the effect of a goaf curtain, and this is achieved by assigning a relatively low permeability to the proposed curtain zones. There is no doubt that the second approach, which is much more convenient, is adopted.

Figure 4.27 depicts the velocity contour for the base model when the curtain is used at the MG corner, where the effect of the curtain can be clearly seen. It is observed that no large scale leakage occurs at the MG corner and the main flow is forced to enter the face. Due to the flow separation at the inner corner, the flow is much more accelerated at the longwall control and communication panel area and the walkway until it passes the shearer. For the rest of the face, the flow pattern is the same as the base model.
Figure 4.28 shows the velocity vectors at the MG corner. It can be seen that though the curtain succeeds in stopping large scale air leakage and the majority of flow is constrained to the face space, there is a small amount of airflow trying to penetrate into the goaf immediately after the curtain ends. The penetration process can be more clearly observed in Figure 4.29 where the airflow streamlines from the MG inlet are illustrated. There is no doubt that a better result can be obtained if the curtain is extended further along the face.

Figure 4.28 Velocity vectors at the MG corner with the application of goaf curtain

Figure 4.29 Effect of goaf curtain on airflow streamlines
The model has revealed that the air leakage can be substantially reduced with the application of a goaf curtain; however, from the perspective of field practice, the effect of goaf curtain has been exaggerated to some degree by the model predicted results. In a practical longwall face, the condition at the MG corner is more complicated, and this may lead to failure of the curtain installation or the curtain length most being as long as it in the model, small gaps may also exist at connections to the rib and floor. Therefore, a small quantity of air leakage is expected in the field implementation of a goaf curtain.

4.5.4 Impact of goaf compaction status

It can be seen from the base model results that the goaf is tightly compacted, and the main air exchange between face and goaf occurs at both the MG and TG corners. Meanwhile, the majority of leaked airflow is able to flow back to the face. However, the occurrence of relatively hard roof is quite common in some mining areas where the longwall roof usually encounters caving problem. To investigate the face flow characteristics under relatively hard roof condition, the overall permeability in the goaf is increased to some extent, representing a less compacted goaf.

Figure 4.30 illustrates the velocity distribution on the surface 2 m above the floor for a less compacted goaf. It can be seen that the overall velocity decreases significantly along the face owing to the increased leakage to the goaf, and the decreasing trend is more distinct in upstream of the MG drum where the velocity magnitude drops to around 1 m/s. The range of air ingression into the goaf expands greatly when compared with the base model results. Meanwhile, the formation of a strip of airflow leakage is observed immediately behind the chocks, and this is better demonstrated by the airflow streamlines illustrated in Figure 4.31. As a certain amount of leaked airflow fails to flow back to the face, the acceleration of flow in the downstream of the shearer body fades away, and an approximately evenly distributed flow field can be observed in further downstream along the face until the flow reaches the TG corner, where the impact of flow separation diminishes in comparison with the base model results.
The streamlines of intake airflow are depicted in Figure 4.31, where it can be seen that the air leakage to the goaf is more severe than the results revealed by the base model. It is noticed that part of the leaked air tries to flow back to the face until the middle of face whilst the majority travels at the back of chocks or is being trapped in deep goaf.
Figure 4.32 shows the airflow streamlines from the MG belt roadway only. The same as in the base model results, the flow is pushed to the inner MG rib in the MG; however, it ingresses into the goaf as the flow enters the face and gradually flows back to the face, indicating the majority of leaked airflow comes from the MG ct.

4.6 Summary

The airflow patterns at longwall faces are complicated owing to the longwall configurations with the involvement of large dimensional equipment. A detailed understanding of airflow field along the longwall face is not achievable using conventional measurement which may affect the normal longwall production as well. Therefore, 3D CFD models were employed in this study to obtain a thorough understanding of the flow characteristics along the longwall face.

Considering the impact of longwall equipment on flow patterns, the models were constructed as close as possible to the actual longwall face geometry by incorporating the key features of the longwall equipment in the models. To investigate the influence of the shearer position and its cutting sequence on the airflow patterns, six CFD models were constructed to represent different operational
scenarios; namely, when the shearer was cutting close to the MG, in the middle of face and close to the TG in both cutting directions.

The MG-TG Case 1 model was taken as the base model and was validated using field monitored velocity profiles where good agreement had been achieved, demonstrating the reliability of the model predicted results. The same boundary conditions were then adopted for the calculation of the other five models.

Generally, the flow characteristics along the longwall face can be summarised as follows:

- Flow separation occurs at both the MG and TG inner corner where the flow boundary changes sharply, leading to the recirculation of airflow at the two corners which is undesirable and should be minimised in the longwall ventilation system;
- Before the MG airflow enters the face, there is serious air leakage to the goaf at the MG outer corner, and the majority of leaked air will flow back to the face at a distance of 20 to 30 m in the face;
- The overall airflow velocity profile changes significantly along the face, with faster airflow above the AFC, and slightly slower in the walkway;
- Airflow is accelerated at the outer TG rib when it enters the TG owing to the occurrence of separation at the inner TG rib, whereas, the flow acceleration at the face entrance is not as significant as that at the TG due to the leakage at the MG corner;
- The shearer position and its cutting direction have a local effect on the airflow pattern along the face, significant flow acceleration is observed in the walkway downstream of the leading drum, especially when the shearer is cutting from the MG to TG;
- In the TG-MG pass, the air leakage to the goaf behind the shearer is more severe than in the MG-TG pass due to the chocks advancing sequence (from TG to MG) and the airflow direction (from MG to TG);
• The rotation of drums affects the airflow patterns around the drums, and the flow around the drums is either slightly accelerated or decelerated depending on the rotating direction;

• The air exchange between the face and goaf mainly occurs at the two ends of the face, and within the first 50 m in the face, the airflow penetrates into the goaf and gradually flows back to the face, further downstream to the face, there is no significant exchange unless the flow passes the shearer, when the flow approaches to the TG, a small quantity of airflow starts to ingress into the goaf and finally returns to the TG from the goaf;

• The use of a goaf curtain at the MG corner can effectively minimise a large quantity of air leakage to the goaf, and the effect can be improved by extending the curtain along the face.

• When the longwall roof encounters caving issues and the goaf is not tightly compacted, the overall velocity along the face is decreased owing to the enhanced air exchange between face and goaf, whilst a certain amount of airflow travels within a strip immediately behind the chocks and is not able to join the main flow until it arrives at the TG corner.
5 MODELLING OF METHANE EMISSION AND FLOW CHARACTERISTICS ON A LONGWALL FACE

5.1 Introduction

In underground coal mines, the mine production is achieved at the longwall face, which is also the largest gas generator. One of the main functions of the ventilation system is to dilute hazardous gas concentrations to safe and acceptable levels. To ensure the required gas level is reached, real time gas monitors are installed at the upper TG corner, in the TG and on the shearer body as well. However, the gas concentration readings at these locations may not represent the highest concentration along the face. As more gas is likely to be liberated as the drums are cutting, the monitors installed far from the drums will not be able to detect the gas concentration around the drums which may be significantly higher. Meanwhile, the installation of gas monitors on the drums is impractical owing to the cutting constraint. Using the longwall models developed in Chapter 4, methane liberation from different sources is modelled to simulate methane accumulation and dispersion characteristics along the entire longwall face and in the vicinity of shearer.

5.2 Determination of methane emission rate at longwall face

To obtain a correct and reliable methane distribution along the longwall face, the rate at which the methane gas is liberated into the working environment needs to be determined in advance. In Chapter 2, various methane emission methods developed across the main coal production countries have been reviewed. It is noted from the review that most of these methods focus on the gas emission from adjacent seams or strata, and the gas emission from the worked seam is considered to be between 50% and 100% coal seam gas content (depending on the desorption rate of coal and face advance rate) without specifying the contribution of different sources, such as gas emitted from coal cut by the drums and coal on the AFC, and emission from the face rib, all of which are necessary inputs for the CFD longwall models. However, based on the mathematical model proposed by Airey (1971), the gas content in the worked
coal seam before cutting can be calculated as well as the gas emission from the cut coal, through which the gas emission from the face rib and the broken coal on the AFC can be determined. Therefore, the determination of the gas emission rate from different sources is described in this section on the basis of Airey’s method.

In Airey’s theory, the coal seam is regarded as a closely packed assembly of “coal lumps”, and gas emits from the coal through the spaces between lumps. The size of coal lump is considered to vary with the distance from coal face to the depth of coal seam. Taking into account the impact of abutment pressure ahead of the face, the size of coal lump is thought to increase progressively from the exposed face rib to deep into the seam, i.e. smaller size close to face rib and larger size further into seam. Thus, the fracture system is better developed near the exposed coal face than the coal deep in the seam, allowing higher gas emission rates at the face rib.

By treating the coal seam as a combination of different sizes of coal lumps, the problem of gas emission from coal seam turns into the integration of gas emission form the coal lumps.

Through a series of derivation processes, the following equations are proposed for the determination of the gas content \( q(x) \) at a position \( x \) in the seam:

For \( x > 0 \),
\[
q(x) = A \exp\left(-\frac{x_0}{r} \exp\left(-\frac{x}{x_0}\right)^{1/3}\right)
\]  \hspace{1cm} (5.1)

For \( x < 0 \),
\[
q(x) = A \exp\left(-\frac{x_0 - x}{r} \right)^{1/3}
\]  \hspace{1cm} (5.2)

where, \( x \) is the position of coal in the seam, it is positive ahead of face, equal to zero at the face line, and negative behind face where coal has been extracted, m;
\( q(x) \) is the gas content at a position \( x \) in the seam, \( m^3/t \);
\( A \) is the coal seam gas content, \( m^3/t \);
\( x_0 \) is a distance constant of the face, m;
\[ r \text{ is the average face advance rate, m/s; } \]
\[ t_0 \text{ is a time constant of degassing, s.} \]

Therefore, the corresponding gas emission \( l(x) \) at position \( x \) in terms of \( \text{m}^3/\text{t} \) in the seam is

\[ l(x) = A - q(x) \quad (5.3) \]

It is known that the face production rate at the longwall face can be expressed as

\[ P = \rho \times L \times H \times r \quad (5.4) \]

where, \( P \) is the face production rate, \( \text{t/s} \);
\[ \rho \text{ is the coal density, } 1.4 \text{ t/m}^3 \text{ (field data); } \]
\[ L \text{ is the face length, } 200 \text{ m for the CFD model; } \]
\[ H \text{ is the cutting height of face, } 3.5 \text{ m for the CFD model.} \]
\[ r \text{ is the average face advance rate, m/s; } \]

Combine Eq. (5.3) and (5.4), the gas emission rate \( k(x) \) at position \( x \) in terms of \( \text{m}^3/\text{s} \) can be determined by

\[ k(x) = l(x) \times P \quad (5.5) \]

As a result, for a given face, the gas emission rate at the exposed face rib can be calculated at \( x=0 \) if the constants and the face advance rate are known. For the determination of gas emission rate along the face rib in the longwall models, it is assumed that, the worked seam gas content is \( 4 \text{ m}^3/\text{t} \); the distance constant and the time constant are \( 4 \text{ m and } 5 \times 10^5 \text{ s (140 h)} \) respectively; the longwall face advance rate is \( 1.16 \times 10^{-4} \text{ m/s (70 m/week)} \). Thus, by substituting these data into Eq. 5.5, the gas emission rate from the exposed face rib is \( 0.1527 \text{ m}^3/\text{s} \), which is also the total emission rate from the entire face rib. However, field data indicated that the gas emission rate from the exposed rib along the face is not uniform but decays with its
exposure time, and the decay is found to follow the hyperbola law, which can be given in the following form (Yu, 1992; Yang, et al., 2009)

\[ q_t = q_0 (1 + t)^{-\beta} \]  

(5.6)

where \( q_t \) and \( q_0 \) are the gas emission rate at \( t \) min and 0 min (at the beginning of exposure), in \( \text{m}^3/(\text{m}^2 \cdot \text{min}) \);

\( \beta \) is the decay coefficient.

It is noted that the decay coefficient \( \beta \) varies with the coal seam geological and mining conditions which can be obtained by regression analysis of field monitored data. According to the field monitoring work carried out by Yang et al. (2009) at a longwall face, a decay coefficient of 0.66 is used in the following calculations. Though the selection of \( \beta \) may not be appropriate for some longwall faces, here it aims to show the methodology which could be used to determine the gas emission rate from the exposed face rib at different positions.

Therefore, up to time \( t \) min, the gas emission from exposed rib can be calculated

\[ Q(t) = \int_0^t q(t)dt = \int_0^t q_0 (1 + t)^{-\beta} dt = \frac{q_0}{1 - \beta} ((1 + t)^{1-\beta} - 1) \]  

(5.7)

It is worth noting that the face rib exposure time can be estimated if the shearer position and its cutting speed are known. Considering the shearer is cutting at a speed of 10 m/min, for the MG-TG Case 1 model, the exposure time for the face rib close to the TG will be around 20 (200/10) minutes. According to Eq. 5.7, the ratio of gas emission in a given time interval (i.e. 1 min) to the total gas emission within a given time \( t \) can be determined. As a result, the gas emission rate in the 1\textsuperscript{st} minute (or for the 1\textsuperscript{st} 10 m immediately behind the shearer) will account for 14.6\% \( \left( \frac{Q(1) - Q(0)}{Q(20)} \right) \) of the total gas emission rate. Similarly, the proportion of gas emission rate in the \( i \textsuperscript{th} \) minute (or for the \( i \textsuperscript{th} \) 10 m range behind the shearer) can be calculated using
\[ \frac{Q(i) - Q(i-1)}{Q(20)} \]. Therefore, knowing the total gas emission rate from the entire exposed face rib (0.1527 m\(^3\)/s), the gas emission rate from the rib at different positions along the face can be determined. It is noticed that when the distance behind the shearer is short, the proportion calculated will be assigned to the rib ahead of the shearer in sequence. Take the MG-TG Case 1 model as an example, the calculated results indicating the determination of gas emission rate from the exposed face rib is shown in Table 5.1. As a result, the gas emission from the face rib ahead of the shearer will be slightly different among the six models, with the highest rate in the MG-TG Case 1 and TG-MG Case 3.

### Table 5.1 Determination of gas emission rate from exposed face rib

<table>
<thead>
<tr>
<th>Model name</th>
<th>Total emission rate (m(^3)/s)</th>
<th>Ratio to the total emission rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>MG-TG Case 1</td>
<td>0.1527</td>
<td>8.8% for the 10 m immediately behind the MG drum; 16.1% for the rest of MG side rib; 8.2% for the first 10 m ahead of the shearer; 7% for the second 10 m ahead of the shearer; 6% for the third 10 m ahead of the shearer; 15% for the fourth to sixth 10 m ahead of the shearer; 12% for the seventh to ninth 10 m ahead of the shearer; 27% for the rest of the TG side rib</td>
</tr>
</tbody>
</table>

Besides the gas emitted from the exposed face rib, the gas contained in the coal continues to be liberated as the drums cut and while the coal is transported on the AFC along the face and on the belt conveyor in the MG. To determine the gas emission rate from the broken coal with relatively smaller sizes, the following equation is employed

\[
V(t) = q(x = 0) \times \left(1 - \exp(-\left(\frac{0.82t}{t_0}\right)^{0.24})\right) \times P
\]  

(5.8)

where \(V(t)\) represents the gas emission rate up to time \(t\), m\(^3\)/s; and the time constant \(t_0\) is chosen as \(1.8 \times 10^5\) s (50 h).

It is noted that a lower value of \(t_0\) is used in Eq. 5.8 as the sizes of broken coal are smaller than the coal lumps in the face before cutting. Meanwhile, the gas emission rate around the drums is calculated as the gas emission from broken coal within the
first second which remains constant as the shearer position changes in the models, and the gas emission from broken coal on the AFC is calculated according to its travel time whilst the emission rate also decays as it travels away from the shearer. Following this principle, the gas emission rate around the drums and on the AFC in all the models is determined and is shown in Table 5.2. Regarding the gas emitted from the MG belt conveyor, a background methane concentration of 0.28% in the MG airflow is used in the models.

Table 5.2 Gas emission rate around the drums and on the AFC

<table>
<thead>
<tr>
<th>Model name</th>
<th>Around the drums (m³/s)</th>
<th>On the AFC (m³/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>MG-TG Case 1</td>
<td>0.0153</td>
<td>0.0108 under and behind the shearer</td>
</tr>
<tr>
<td>MG-TG Case 2</td>
<td>0.0153</td>
<td>0.0108 for the first 30 m under and immediately behind the shearer, 0.0115 for the rest 80 m</td>
</tr>
<tr>
<td>MG-TG Case 3</td>
<td>0.0153</td>
<td>0.0108 for the first 30 m under and immediately behind the shearer, 0.0115 for the following 80 m, 0.0050 for the rest 60 m</td>
</tr>
<tr>
<td>TG-MG Case 1</td>
<td>0.0153</td>
<td>The same as MG-TG Case 1</td>
</tr>
<tr>
<td>TG-MG Case 2</td>
<td>0.0153</td>
<td>The same as MG-TG Case 2</td>
</tr>
<tr>
<td>TG-MG Case 3</td>
<td>0.0153</td>
<td>The same as MG-TG Case 3</td>
</tr>
</tbody>
</table>

In the model, it is assumed that there is no gas bearing strata and coal seams above or under the worked seam, so the gas emission from the goaf is mainly composed of the broken coal left in the goaf. For a typical fully mechanised longwall face, a 95% of recovery ratio is adopted. Considering an immediate goaf with 10 m depth, the emission rate is determined as 0.0092 m³/s with $P$ reduced by 95% in Eq. 5.8.

### 5.3 Modelling of gas flow in the underground ventilation system

Unlike the normal air which is mainly composed of oxygen, nitrogen and carbon dioxide, the underground atmosphere contains a certain amount of other gases, such as methane, carbon monoxide, and nitrogen oxide. The occurrence of these gases can cause toxic and an explosion hazard if not managed properly. Therefore, it is imperative to understand and be able to predict the flow characteristics of the hazardous gases in the underground ventilation system. In this case, the conservation equations describing the convection, diffusion and even the reaction of gases need to be solved, and this is achieved using the species transport model.
Through the solution of convection-diffusion equations for all the chemical species, the local mass fraction of each species can be predicted. And a general form of the conservation equation for the $i$th species can be expressed as (ANSYS, 2010):

$$\frac{\partial (\rho Y_i)}{\partial t} + \nabla \cdot (\rho \vec{v} Y_i) = -\nabla \cdot \vec{J}_i + R_i + S_i$$  \hspace{1cm} (5.9)

Where, $Y_i$ is the mass fraction of the $i$th species;

$\vec{J}_i$ is the mass diffusion flux of the $i$th species;

$R_i$ is the net rate of production of the $i$th species by chemical reaction;

$S_i$ is the rate of creation by addition from the dispersed phase plus any user defined sources.

For a gas mixture containing $N$ species, $N-1$ similar equations to Eq. 5.9 need to be solved for the corresponding $N-1$ species. And the $N$th mass fraction is calculated by one minus the sum of the $N-1$ obtained mass fractions. It is noted that the $N$th species should be chosen as the species with the overall largest mass fraction to minimise the numerical error (ANSYS, 2010). For the purpose of this study, only three main components in the mine air are modelled, i.e., methane, oxygen and nitrogen.

The mass diffusion flux in the turbulent flow can be given as (ANSYS, 2010):

$$\vec{J}_i = -\left(\rho D_{i,m} + \frac{\mu_i}{Sc_i}\right) \nabla Y_i - D_{r,i} \frac{\nabla T}{T}$$  \hspace{1cm} (5.10)

where $\mu_i$ is the turbulent viscosity and $Sc_i$ is the turbulent Schmidt number.

5.4 Base models results

Using the ventilation system and flow rate illustrated in Chapter 4, methane flow characteristics along the longwall face have been investigated with 4 m$^3$/t of coal seam gas content. Meanwhile, comparisons are made among the models to
understand the impact of cutting sequence and shearer positions on the methane flow behaviour along the longwall face and at the TG corner.

5.4.1 Overall methane distribution

Figure 5.1 depicts the overall methane concentration distribution when the shearer is cutting at different positions along the longwall face. It can be seen that there is no difference in the methane distribution in the MG among the six models, the methane contained in the airflow from the belt roadway is instantly diluted by airflow from the MG cut through and diverted towards the inner MG rib side. As the flow enters the face, significant variation of methane distribution can be observed along the face and in the goaf. At the inner MG corner, where the flow separation occurs, the methane concentration is relatively higher within the recirculation zone. It is noted that this effect gradually diminishes as the shearer cuts further towards the TG and starts to recover as the shearer cuts back from the TG to the MG. The main cause to this phenomenon can be attributed to the decay of gas emission from the exposed face rib.

The diffusion of methane from the face rib to the walkway can also be observed in Figure 5.1. In the MG-TG pass, as the shearer cuts from the MG to the TG, the general methane concentration increases along the face, especially in the vicinity of TG drum where it has increased from around 0.74% to 1.23%. In the TG-MG pass, the methane concentration decreases correspondingly along the face as the shearer cuts back from the TG to the MG. As a result, the methane concentration is highest when the shearer is cutting at the tail end of the face. In both cutting directions, the methane concentration is relatively low along the face for the first half of the face, and it slowly increases as the flow travels to the TG in the second half of the face. The methane flow pattern in the immediate goaf changes significantly as the shearer is cutting at different positions, especially at the tail end of the face. At the MG side of goaf, the methane is sufficiently diluted by the penetrated airflow, which is consistent among the six models. The accumulation of methane mainly occurs at the middle and TG side of the goaf where the goaf is more compacted and the airflow is weak. The impact of shearer position on the goaf methane flow behaviour can be
clearly observed. As the shearer cuts to the middle and the TG, the air ingress to the goaf is increased at the shearer position and the methane is diluted slightly in the vicinity, flushing the methane to the deep goaf at the same time. This effect is more obvious as the shearer approaches to the TG. Therefore, unlike the gradual methane accumulation along the face as the shearer moves towards the TG, the methane concentration at the upper TG corner decreases in the MG-TG pass.
As the ventilation enters the TG, the effect of flow separation on the methane distribution at the inner TG corner is significant. The majority of methane gas liberated at the face is involved in the flow recirculation zone, characterised with high methane concentration, and its range extends greatly as the shearer cuts from the MG towards the TG. Due to the decay of gas emission from the exposed face rib, the range variation of the high methane concentration zone in the TG-MG pass is not as significant as that in the MG-TG pass; however, in both passes, the maximum methane concentration can reach a stable value of approximately 0.75% at the inner
TG corner (as indicated in Figure 5.1). As the flow travels further downstream in the TG, the methane gas gradually diffuses from the high methane concentration region (close to the inner TG rib) to the low concentration region (close to the outer TG rib), and the methane distribution in the TG will eventually become uniform.

It can also be observed from Figure 5.1 that the source of methane gas leading to accumulation at the intersection of face and TG can be identified, i.e. methane liberated from the longwall face will accumulate at the inner TG corner where the flow separation occurs, whilst the methane emitted from the goaf is more likely to accumulate at the upper TG corner. Therefore, it is recommended that, for single coal seam extraction where the methane is mainly emitted from the coal face (the modelled condition), attention should be paid to the methane accumulation at the inner TG corner, especially when the shearer is approaching the TG; for multi-seam mining conditions where a large amount of methane may enter the face from the goaf, control measures should be carried out to ensure the methane level at the upper TG corner is diluted below the statutory requirement. Two independent real time gas monitors are suggested to be installed at the two positions.

5.4.2 Methane distribution around the shearer

Figure 5.2 illustrates the methane concentration distribution around the shearer where the methane concentration on both the shearer and the face ribs are shown, and the 1% methane concentration contour is also illustrated in Figure 5.2. It can be seen that the methane concentration varies periodically as the shearer cuts at different positions in the two cutting directions, i.e. the methane concentration gradually builds up in the vicinity of the shearer as it cuts from the MG to the TG, and diminishes as the shearer cuts back from the TG to the MG.

It is worth noting that the methane concentration around the TG drum is significantly higher than it is around the MG drum, regardless of the cutting sequence. It is acknowledged that there are two main factors resulting in this kind of methane accumulation mechanism. Firstly, as can be seen from Figure 5.2, in the MG-TG pass, the shearer travels downwind of the airflow, the fresh rib generated by the TG
drum will act like a curtain, blocking the airflow to some extent. Meanwhile, the leading TG drum, which accomplishes 60% of the longwall production, will liberate a large amount of methane from the newly exposed rib, which is likely to accumulate in the vicinity of the TG drum owing to the blockage of the rib against the flow direction. However, in the TG-MG pass, though the leading drum (the MG drum) extracts 60% of coal, the blockage of new rib in front of the MG drum is not as significant as that in the MG-TG pass, and the methane emitted downstream of the MG drum is diluted by the upstream airflow, after which the methane will gradually accumulate as the flow approaches to the TG instead of gathering around the leading drum as in the MG-TG pass. Secondly, the obstruction of the shearer body plays an important role in the process. Irrespective of the cutting sequence, the airflow downstream of the shearer body will be retarded to some degree, as well as the airflow downstream of the TG drum, especially in the TG-MG pass where the TG drum cuts the lower section of the face and the effect of shearer obstruction is thus more significant.

![Diagram](image)

(a) MG-TG Case 1  
(b) MG-TG Case 2
It is also observed that the rotation of drums affects the methane flow in the vicinity of drums though the effect is not comparable to the obstruction of the shearer body. It is known that the TG drum rotates anticlockwise and the MG drum rotates clockwise if looking from the walkway. Taking the TG drum as an example, the rotation of drum will generate a positive and negative motion to the airflow at the top and bottom of the drum respectively. Therefore, the flow is accelerated at the top of the drum while it is retarded at the bottom, and the rotation also brings a small amount of methane to the sluggish airflow at the lower section of the TG drum, generating a favourable environment for the methane to accumulate. As a result, the
methane concentration is slightly higher at the lower section of the TG drum. On the contrary, a higher concentration area can be observed at the upper section of the MG drum rather than the lower section.

Accompanied by the significant variation of methane concentration around the drums, the methane concentration on the shearer body also changes greatly as the shearer moves from the MG to the TG and reversely. Generally, the change of methane concentration follows the same trend as it does around the drums, i.e. increases gradually as the shearer cuts from the MG to the TG and decreases correspondingly from the TG to the MG. For all the six cases modelled, the methane distribution varies across the shearer body. The methane concentration is slightly higher at the face rib side owing to the diffusion of the methane from the face rib, and it gradually decreases to the walkway side. Due to methane emitted from the broken coal on the AFC, the methane level is slightly increased on the shearer body in front of the spill plate on the walkway side, as indicated in Figure 5.2.

From a perspective of gas management along the longwall face, the methane level around the drums should be closely monitored. However, due to the constraints of field production practice, the installation of a real time gas monitor on the drum is not available, thus it has to be installed on the tail end of the shearer body at most longwall faces. As can be seen from Figure 5.2, the reading obtained from a shearer monitor can be approximately two or three times lower than the gas concentration around the TG drum. Therefore, it is vital to investigate the difference between the methane concentration at the tail end of the shearer body and the TG drum, through which the methane level at the TG drum can be estimated using the monitored data during field practice.

5.4.3 Methane distribution across the face

Figure 5.3 shows the methane concentration distribution at four different positions across the face, and the height investigated is 2 m above the floor. According to the face geometry, the variation of methane concentration can be divided into three sections, i.e., the face space, space between chock legs and the immediate goaf. It
can be seen that the methane distribution presents significantly distinct characteristics among the three sections. At the face area, the methane concentration decreases sharply from the face rib towards the walkway, and the decreasing trend slows down as it moves to the space between chock legs, where the variation trend diverts significantly. At the MG corner and the middle of shearer body, the concentration decreases with different gradients between chock legs and becomes flat in the goaf. While the concentration increases slightly at the middle of face and TG corner.

The gradual build up of methane from MG to TG along the face can be observed clearly in Figure 5.3. It is noted that the results are based on the MG-TG Case 1.

![Figure 5.3 Methane concentration distribution cross the face at 2 m above floor](image)

5.5 Methane flow characteristics under various operating conditions

It is known that the methane flow at a longwall face is quite a complex process, which is not only closely related to the mining conditions but also relies on the
ventilation system adopted. The distribution of methane on a longwall face, where the coal extraction is carried out in a single seam with 4 m³/t gas content, is investigated in section 5.4 under normal ventilation conditions. However, for coal seams with different gas content and varying occurrences of coal measure rocks, the mining methods and the ventilation system may vary significantly in terms of panel layout and flow rate, and subsequently the corresponding methane flow behaviour at longwall faces. Thus, it is of great importance to investigate the methane flow characteristics under various operating conditions for the purpose of providing fundamental understanding and guidance to longwall face methane management.

5.5.1 Impact of ventilation

For single seam extraction, the majority of methane is liberated from the exposed face rib, and the overall methane level is demonstrated to be the highest as the shearer approaches to the TG. Therefore, the MG-TG Case 3 is chosen to investigate the effect of ventilation on methane flow behaviour on the longwall face. It is important to point out that the impact of ventilation is evaluated by reducing and increasing the flow rate to 35 m³/s and 55 m³/s respectively, which is achieved by adjusting 10 m³/s of airflow to the MG cut through only, whilst all the other conditions are kept the same as the base model.

The contour of methane concentration at 2 m above the floor is illustrated in Figure 5.4, where the same colour scale is used for the convenience of comparison with base model result (refer to Figure 5.1.c). It is observed from Figure 5.4 that the general methane flow patterns along the face and in the goaf are similar for the three cases; however, they vary from each other in terms of methane levels. When the flow rate is reduced by 10 m³/s, the range filled with high concentration of methane expands significantly in the vicinity of the shearer and further downstream, where the methane level is increased by approximately 0.3% to 0.5% compared with the result of the base model and the methane level at the inner TG corner is increased to around 1%. The methane level also increased in the goaf, with a maximum concentration of 0.74%. Due to the relatively weak flow along the face, the resistance to prevent goaf methane entering the face is reduced to some degree, resulting in a small amount of
goaf methane emitting into the face. Combined with the diffusion of methane from the face rib, the overall methane level in the walkway is increased as well. In comparison, when the flow rate is increased by 10 m$^3$/s, both the methane liberated from the face rib and the goaf are better diluted and flushed to the rib and deep goaf respectively. In this case, the mix of methane from the two sources at the face will be retarded further downstream of the face. Depending on the flow rate increase, the methane emitted from the face and goaf may not be able to blend until the flow reaches the TG corner.

It is important to note that, at the upper TG corner, the methane level does not drop significantly even when the face flow rate is increased by 20 m$^3$/s. This is actually a disadvantage of the “U” ventilation scheme which is demonstrated by the two modelled scenarios. The “U” ventilation scheme is not effective in diluting the methane accumulated at the upper TG corner, which is mainly liberated from the goaf.

![Diagram](image)

(a) Reduces the flow rate to 35 m$^3$/s

(b) Increases the flow rate to 55 m$^3$/s

Figure 5.4 Ventilation impact on the methane distribution – 2 m above floor
It is worth noting that the methane flow behaviour along the face and in the goaf is affected by the change of airflow rate, and the influence to both the face methane and the goaf methane is consistent, i.e. the methane level at different positions indicating the methane sources is reduced as the increase of flow rate at face.

The variation of methane concentration around the shearer is depicted in Figure 5.5, in which the iso-surface of 1% methane concentration is also illustrated. It can be seen that the methane level in the vicinity of the shearer is greatly increased (reduced) when the flow rate is reduced (increased).

![Variation of methane concentration around the shearer](image)

(a) Reduces the flow rate to 35 m³/s  \hspace{1cm} (b) Increases the flow rate to 55 m³/s

Figure 5.5 Variation of methane concentration around the shearer

It is noticed from Figure 5.5 that when the flow rate is reduced to 35 m³/s, the entire TG drum and the majority of the MG drum are surrounded by airflow composed of more than 1% of methane, as well as zones further downstream the shearer where the entire face rib is also covered by methane levels greater than 1%. In addition, the methane level is increased upstream of the MG drum. A similar changing trend can also be observed on the shearer body. While the methane emitted around the shearer is adequately diluted when the flow rate is increased to 55 m³/s, the high methane concentration is confined to the zone ahead of the TG drum and immediately downstream of its ranging arm.
Therefore, it can be concluded from Figure 5.4 and Figure 5.5 that the methane distribution on a longwall face is highly dependent on the flow rate provided. It is vital to ensure that a stable ventilation system is maintained on the longwall face. Meanwhile, the “U” ventilation scheme is not recommended when the goaf methane emission accounts for the major methane source on a longwall face.

5.5.2 Impact of coal seam gas content

In the base models, the methane flow characteristics have been investigated with the assumption of extracting a single seam with relatively low gas content of 4 m³/t. The model results demonstrate that 45 m³/s of airflow is capable of diluting the methane level at required positions under the statutory requirement. However, the methane content in coal seams is rarely constant, depending upon pre-drainage effect, local geological conditions and the mining activities conducted in overlying or underlying seams. In some cases, the change of local geological conditions may result in a great increase in methane emission to the working environment. Therefore, in this section, it is assumed that the longwall face is extracting through an abnormal methane emission section, where the methane content in the coal seam has increased to 8 m³/t. Taking the MG-TG Case 3 as the base model, the methane flow characteristics on the longwall face is investigated with the ventilation system unchanged. It should be noted that, according to the method described in section 5.2, the methane emission rate is two times the rate used in the base model.

The methane concentration distribution at 2 m above the floor is illustrated in Figure 5.6, where similar methane distribution patterns to the base model results can be observed. However, as can be seen from the scale, the general methane concentration is greatly increased, especially in the vicinity of drums, where the maximum methane concentration can be as high as 3%. Meanwhile, due to the use of scale up to 3%, the emission of methane from the exposed face rib and its diffusion in the first two thirds of the longwall face is not clearly depicted in Figure 5.6. In the last third of the face, the methane gradually builds up and its concentration has exceeded 1% along the face rib; directly above the shearer body, the methane concentration also rises to 0.9%. Downstream of the shearer, there is still a high methane concentration strip.
where the concentration can be above 2%. As the flow enters the TG, the same methane distribution pattern as the other models can be observed. The occurrence of flow separation as the flow enters the TG is capable of trapping a large amount of methane gas at the inner TG corner, and in this case the methane level reaches to around 1.5%. As inferred before, the trapped methane is mainly from the longwall face rather than the goaf. At the upper TG corner, which constitutes an important goaf gas discharge to the airflow, the methane concentration increases to approximately 0.75%. Eventually, the variation of methane concentration diminishes and a return airflow containing 0.9% of methane is generated as the flow travels further downstream in the TG.

![Methane distribution on the surface 2 m above floor - 8 m³/t gas content](image)

The distribution of methane concentration around the shearer is shown in Figure 5.7, where both the 1% and 2% methane concentration iso-surfaces are illustrated. Compared with Figure 5.2c, the spatial distribution 1% iso-surface is significantly enlarged and extended further to the walkway side, especially in the vicinity of the floor and roof level. It is observed that, both upstream and downstream of the shearer, the entire face rib is enclosed by airflow containing more than 1% of methane. Upstream of the shearer, it can be easily seen that there is significant methane accumulation above the AFC and the 1% iso-surface is approaching to the top of the shearer body. This can be attributed to the fact that the AFC is approaching to its maximum load in the MG-TG Case 3, combined with obstruction of the shearer body and spill plate, the effect of methane accumulation above the AFC is much more significant when the gas emission is greatly increased. It is also noted that the entire
MG drum and the majority of its ranging arm are enveloped with higher than 1% of methane-air gaseous mixture, as well as the TG drum and its ranging arm. At the roof level, the expansion of the width of the 1% iso-surface can be observed, which indicates the diffusion and accumulation of methane emitted from the face rib; while the width shrinks slowly down to the middle of the face. It is analysed that the reduced flow velocity in the vicinity of roof is the main cause leading to this kind of spatial distribution patterns. The 2% iso-surface shows a similar pattern to the 1% iso-surface of the base model, indicating the methane concentration in the vicinity of face rib and drums is approximately doubled when the gas emission rate is doubled.

![Figure 5.7 Methane distribution around the shearer with 8 m³/t gas content](image)

From a perspective of field methane monitoring around the shearer, it is worth noting that the methane concentration at the tail end of the shearer body is still less than 1%, not exceeding the requirement for power outage. Unfortunately, the methane level at the inner TG corner has exceeded the limit, thus, the shearer would not have been able to cut to the position without frequent power outages or adoption of other control measures. Therefore, it is vital to be aware of the difference between the monitored and the actual maximum methane concentration which may be experienced during longwall production. It is obvious that the capability of the original flow rate (45 m³/s) is insufficient to deal with such a significant increase in gas emission rate along the face. In other words, the current “U” ventilation system
ventilated with 45 m$^3$/s of airflow is not capable of extracting a single coal seam with 8 m$^3$/t of methane content. Effective pre drainage strategies or ventilation system enhancement measures should be put into effect to ensure the safety of longwall production.

5.5.3 Impact of adjacent gas bearing strata

It is known from the above model results that, regardless of the flow rate at the face and the coal seam gas content, methane gas is likely to accumulate around the drums and at the inner TG corner. Methane concentration at the upper TG corner is relatively lower than it at the inner TG corner, and around the drums. It is worth noting that this conclusion is applicable to single seam extraction where the gas emission from adjacent gas bearing strata is not taken into account. In some cases, the sandstone or coal seams in the overlying/underlying strata may contain large amount of methane gas. Once disturbed by the mining activity, a certain amount of methane is likely to enter the working face, adding a significant methane source to the face methane emission. Depending on the distance to the working seam and the intensity of mining activities in the working seam, the amount of methane capable of migrating into the face varies. Therefore, in this section, the impact of methane emission from adjacent gas bearing strata is investigated. The major flow paths for the adjacent methane entering the face are the fractures generated in the goaf as the face retreats, and eventually, this part of the methane flows into the goaf before entering the face, thus, the methane emission from the goaf is assumed to have increased by four times the original goaf methane emission rate, and methane emission along the face is maintained as the same as that in the base model.

Taking both the MG-TG Case 1 and the MG-TG Case 3 as base models, the impact of methane emission from adjacent strata is investigated along the face. Figure 5.8 illustrates the methane concentration contour 2 m above the floor when the shearer is cutting at the MG side. It can be seen that the general methane distribution pattern is similar to that of the base model; however, the methane concentration increases significantly in the goaf, especially in the second and last thirds of the goaf, where the maximum concentration reaches 2.34%. In the first third of the goaf, where there
is significant air leakage to the goaf, the methane concentration is kept at a low level as it is in the base model, and the methane emitted in the area is flushed to the middle section. While in the rest of the goaf, the build-up of methane gradually appears and the goaf methane is trying to diffuse into the face as it accumulates in the deep goaf. It is observed that the methane concentration between the chocks’ legs can be as high as 1%. Meanwhile, as the flow approaches the TG, the goaf methane is slightly diluted by the leaked air to the goaf, pushing the methane to the deep goaf, where a semi-oval shaped zone of high methane concentration is formed.

![Figure 5.8 Methane concentration distribution at 2 m above floor - MG-TG Case 1](image)

At the intersection of face and the TG, it is observed that the methane concentration at the upper TG corner is much higher than it at the inner TG corner, implying the necessity of monitoring the methane concentration at the upper TG corner. The methane distribution at the TG corner also demonstrates the inference that the major methane source at the upper TG corner is the goaf methane while at the inner TG corner the methane mainly comes from the longwall face. As a result, it is possible to identify the methane sources leading to power outages in the actual production, and subsequently implement corresponding methane control strategies.

Figure 5.9 depicts the methane distribution 2 m above the floor when the shearer is cutting near the TG. It is observed that the overall methane distribution in the goaf is similar to the results of MG-TG Case 1, and the methane concentration is slightly lower than in the MG-TG Case 1, which is the same as the comparison between the
two base models. The dilution of methane at the tail end of the goaf is more significant than that in the MG-TG Case 1 owing to the increased air leakage to the goaf in the MG-TG Case 3. Consequently, the methane concentration at the upper TG corner is slightly lower. Meanwhile, the implication mentioned before is also applicable to the MG-TG Case 3 and all the other base cases described in section 5.4.

![Figure 5.9 Methane concentration distribution at 2 m above floor - MG-TG Case 3](image)

The spatial distribution of the 1% methane concentration iso-surface for MG-TG Case 1 and Case 3 are illustrated in Figure 5.10 and Figure 5.11 respectively.

![Figure 5.10 Spatial distribution of the 1% methane concentration iso-surface - MG-TG Case 1](image)
Figure 5.11 Spatial distribution of the 1% methane concentration iso-surface - MG-TG Case 3

It can be seen from Figure 5.10 and Figure 5.11 that the methane concentration exceeds 1% at the upper TG corner, and the range reduces as the shearer cuts from the MG towards the TG. Meanwhile, the 1% iso-surface is flushed backwards to the deep goaf when the shearer is approaching the tail end of the face. While at the inner TG corner, the methane concentration is lower than 1% in both cases, even though it is approaching 1% for the MG-TG Case 3. It is also observed that the 1% iso-surface is not straight up, but it leans forward to the TG at the roof level, indicating that real time methane monitors should be located close to the roof rather than at lower positions. For the methane distribution along the face, there is no significant variation when compared with the corresponding base models.

Therefore, it can be concluded that increased methane emission from the adjacent gas bearing strata has a significant impact on the methane distribution in the immediate goaf and at the upper TG corner of the face. For the case investigated, the goaf methane has a minor influence on the methane distribution along the face and in the vicinity of the shearer; however, the tendency of the goaf methane to enter the
working face is identified. This tendency may be enhanced if the emission rate is increased significantly in the goaf.

5.5.4 Impact of the TG cut through

It has been demonstrated in section 5.5.3 that increased goaf methane emission significantly affects the methane concentration at the upper TG corner. The methane concentration at the upper TG corner may exceed the statutory limit frequently if no effective control measures are implemented, which will substantially affect the normal longwall production. As is known from section 5.5.1, the variation of face flow rate has a major effect on the methane distribution along the longwall face and at the inner TG corner; however, its impact on the methane level at the upper TG corner is minor. Therefore, rather than adjusting the flow rate to the face, the ventilation scheme is modified slightly aiming at mitigating methane accumulation at the upper TG corner. This is achieved by adding a cut through on the tail end of the goaf, which can be seen in Figure 5.12. As a result, the “U” ventilation scheme is transformed into the “U+L” ventilation scheme. It is noted that this section is actually an extension of section 5.5.3, which aims at investigating the feasibility of solving the methane accumulation issue from the perspective of ventilation scheme irrespective of the face flow rate.

![Figure 5.12 Methane concentration contour at 2 m above floor with 17.5 m³/s flow rate through the TG cut through](image)
It is known that the methane level at the upper TG corner is higher in the MG-TG Case 1 than in the MG-TG Case 3, thus the MG-TG Case 1 model is used as the base model, and the calculations are conducted on the basis of results illustrated in Figure 5.8. As can be seen in Figure 5.12, the TG cut through is located behind the face and is directly connected to the goaf, so it will be affected by the collapse of roof in the goaf, and the space within the cut through may be partially filled with broken rocks, blocking a large amount of airflow passing through. Considering the obstruction of rock blocks in the TG cut through, the flow rate travelling through the TG cut through is assumed to be 17.5 m$^3$/s, 10.5 m$^3$/s and 3.5 m$^3$/s, representing three different roof caving conditions in the cut through, and its effect on the methane level at the upper TG corner is investigated through three independent cases.

Figure 5.12 illustrates the methane concentration contour 2 m above the floor when the flow rate at the TG cut through is 17.5 m$^3$/s. Compared with Figure 5.8, it can be seen that the use of “U+L” ventilation scheme does not affect the general methane distribution patterns; its impact is confined to the tail end of the face and goaf and subsequently to the TG airflow. At the upper TG corner, the methane concentration decreases significantly as a portion of flow travels to the TG cut through behind the face, the goaf methane which used to accumulate at the upper TG corner is flushed backwards to the deep goaf; meanwhile, the amount of goaf methane entering the TG is greatly reduced, and the methane emitted from the longwall face constitutes the major methane source in the main return airflow. However, due to the split of main flow at the TG corner, the methane level in the TG does not decrease.

When the flow rate passing through the cut through is reduced to 10.5 m$^3$/s, the methane distribution on the surface 2 m above the floor is depicted in Figure 5.13. It can be observed that, as the partially collapsed TG in the goaf comprises the main path for the split flow to migrate towards the cut through, the methane gas is also well diluted at the upper TG corner and the magnitude of concentration has dropped to a lower level than at the inner TG corner. Consequently, the decrease of split flow rate leads to the slightly increased methane level at the cut through. It is also noticed from Figure 5.13 that there is a significant tendency for the goaf methane to migrate towards the upper TG corner.
Figure 5.13 Methane concentration contour at 2 m above floor with 10.5 m$^3$/s flow rate through the TG cut through

Figure 5.14 illustrates the results when the TG cut through is nearly blocked but there is still a small amount of airflow being able to pass through. Slightly decreased methane level at the upper TG corner is observed; however, it is still significantly higher than at the inner TG corner, which is far beyond the results obtained in the previous two cases. Meanwhile, it is noticed that the methane level in the cut through is also significantly higher than in the previous two cases.

Figure 5.14 Methane concentration contour at 2 m above floor with 3.5 m$^3$/s flow rate through the TG cut through

Figure 5.15 and Figure 5.17 respectively show the spatial distribution of the 1% methane concentration iso-surface for the three corresponding cases. It can be seen that there is a significant propensity for goaf methane migration towards the upper
TG corner as the flow rate through the cut through decreases. No significant variation on the face methane concentration is observed.

From the three cases investigated, it can be concluded that the methane accumulated at the upper TG corner can be effectively diluted with the use of “U+L” ventilation scheme, especially when the flow rate through the TG cut through is greater than 10.5 m$^3$/s. The model results also indicate that it is important to keep the integrity of the TG cut through and ensure the validity of “U+L” ventilation scheme; otherwise, it degenerates to the “U” ventilation scheme, and the goaf methane accumulates at the upper TG corner again. Based on the model results, it can also be speculated that, as the face advances and the TG cut through falls further into the goaf the flow resistance will be greatly increased, the dilution of methane at the upper TG corner will be weakened. As a result, there should be an optimal interval between cut throughs in terms of exerting the best performance of the “U+L” ventilation scheme.

Figure 5.15 Spatial distribution of 1% methane concentration iso-surface with 17.5 m$^3$/s flow rate through the TG cut through
Figure 5.16 Spatial distribution of 1% methane concentration iso-surface with 10.5 m³/s flow rate through the TG cut through

Figure 5.17 Spatial distribution of 1% methane concentration iso-surface with 3.5 m³/s flow rate through the TG cut through
5.5.5 Impact of drum sprays

It is known that not only the cutting picks and also sprays are installed on the shearer drums. The initial intention of installing sprays on the drums is to suppress the coal dust generated in the process of coal cutting, when the methane gas is liberated simultaneously. As a result, the operation of sprays will undoubtedly affect the local airflow patterns and subsequently the methane distribution patterns. However, due to the limitations of field conditions, the sprays’ impact on the methane flow patterns cannot be investigated through field monitoring. Therefore, in this section, the CFD models are employed again to probe into the interactions of sprays and the methane distribution patterns in the vicinity of the drums.

In this section, the MG-TG Case 3 is used as the base model, in which the sprays installed on both drums are switched on (refer to Figure 4.1c for the layout of the drum sprays). It is noted that a total number of 40 sprays are installed on each drum, and the sprays are operated with a velocity of 3 m/s at 30 degrees outwards the tangential direction of the drum which is consistent with the drum’s rotating direction. The operation of sprays is modelled as air sprays for simplification which is achieved by setting the outer face as velocity inlet, allowing air to be ejected from the sprays.

The methane concentration distribution around the shearer is shown in Figure 5.18, where the 1% methane concentration iso-surface is also illustrated. Compared with the base model result, it can be seen that the impact of sprays on the methane distribution is confined to the areas near the face rib, especially in the vicinity of and a short distance downstream of the drums, where the methane level decreases slightly. The influence of the sprays on the methane level at the shearer body and the ranging arms is marginal, indicating these areas are beyond the range of the drums’ influence.
Closer views of the methane distribution around both the TG drum and the MG drum are depicted in Figure 5.19 and Figure 5.20 respectively. It is observed that, when the drum sprays are off, the majority of the TG drum is enclosed by the 1% methane concentration iso-surface, except the higher section of drum on the walkway side; while with the operation of the sprays, the majority of the TG drum is surrounded by airflow with methane concentration less than 1%, and the 1% iso-surface is pushed towards the face rib. Meanwhile, there is significant decrease in methane concentration downstream of the TG drum at the drum’s horizontal axis level. However, areas at the bottom of the TG drum and its downstream are beyond the influence of the sprays, where the methane concentration does not exhibit any significant dropping or rising tendency. The impact of the MG drum sprays on the local methane distribution can be observed in Figure 5.20. It can be seen that the MG drum sprays have a major effect on the methane distribution around the MG drum. The range enveloped by the 1% methane concentration iso-surface is greatly reduced, though it is not as large as that around the TG drum.
Figure 5.19 Methane concentration distribution around the TG drum (left: base model; right: with drum sprays on)

Figure 5.20 Methane concentration distribution around the MG drum (left: base model; right: with drum sprays on)

Figure 5.21 to Figure 5.23 illustrate the variation of methane concentration around the shearer at three different levels above the floor, where both the results with and without the operation of sprays are shown for the convenience of comparison.
At the 1.5 m level, which is approaching the top of the shearer body, there is a significant reduction in methane level downstream of the TG drum, and the red colour which represents higher methane level approximately fades away. It is also observed that the methane level also decreases immediately behind the MG drum, but its range of influence is not as large as the TG drum at the level investigated. However, in the upstream of both drums, it is noticed that the methane level does not indicate any descending trend. In the face retreating direction, as noted before, the influence of drum sprays on the methane level backwards of the shearer is negligible.

At the 2 m and 2.5 m level, a similar variation trend of the methane concentration to the 1.5 m level can be observed in Figure 5.22 and Figure 5.23 respectively. Meanwhile, it is noticed that the impact of the MG drum sprays in its downstream can also be seen near the top of the MG drum, and its impact vanishes at levels above it.

Figure 5.21 Methane concentration distribution at 1.5 m above floor
From above model results, it can be concluded that, with the operation of drum sprays, the methane level in the vicinity of drums and downstream is greatly reduced,
and no significant rising tendency of methane concentration is observed, both of which indicate the operation of drum sprays has important effects on the methane accumulation around the drums. Therefore, the drums’ sprays should be on as the shearer cuts the coal.

5.5.6 Impact of curtain on the MG side

To reduce the airflow leakage into the goaf at the MG corner, a curtain is usually installed behind the MG chocks. In Chapter 4, the impact of the MG curtain on the airflow patterns was investigated, and it was concluded that the use of MG curtain could effectively reduce the leakage. As the methane distribution is highly dependent on the airflow patterns, it is important to investigate the influence of MG curtain on the corresponding methane distribution characteristics.

In this section, both the MG-TG Case 1 and Case 3 are used as the base model, the curtain employed owns the same parameter as the one used in Chapter 4, which is 7 m long and extends from the outer MG rib to the fourth chock in the face. Figure 5.24 indicates the overall methane distribution with the curtain used at the MG corner. Compared with the base models results, the general methane distribution pattern remains the same, except in the first third of the longwall face, where the methane diffusion along the face is confined to the vicinity of face rib owing to the increase of flow rate along the face. Meanwhile, it is observed that, for both cases, there is no significant decrease in the methane concentration along the face; however, the high methane concentration region expands slightly in the goaf. The region affected by the flow separation at the inner MG corner becomes long and narrow, which is more clearly illustrated in Figure 5.24a. While at the inner TG corner, the methane level and the range affected by the flow separation do not change significantly.

Therefore, it can be concluded from the model results that, the impact of MG curtains on the methane distribution along the face is found to be minor, and its impact is confined to the first third of the longwall face, where the airflow patterns are greatly affected by the MG curtain.
5.6 Summary

On the basis of the CFD models developed in Chapter 4, the methane gas distribution along the face and in the immediate goaf was investigated. The longwall extraction was assumed to be conducted in a single coal seam with a methane content of 4 m$^3$/t, and the weekly face advance rate was taken as 70 m. According to Airey’s theory, the gas emission rate from various sources of the face was calculated, i.e., methane emitted from face rib, drum cutting, AFC, belt conveyor and the goaf, which was then used as input for the models. Considering the variation of the shearer position and the cutting sequence among the base models, the decay of methane emission was taken into account as well.
The model calculations were carried out using the species transport model without reactions. And the ventilation system used in the base models was the “U” ventilation scheme with a flow rate of 45 m$^3$/s. The results of base models indicated that:

- There is significant methane accumulation at the inner MG corner where the flow separation occurs, but the methane concentration is low compared with that in the vicinity of drums;
- The methane concentration around the drums gradually increases as the shearer cuts from the MG to TG, and decreases as the shearer cuts back towards the MG, as well as the methane concentration at the inner TG corner, however, at the upper TG corner, it changes reversely;
- The majority of methane trapped in the inner TG corner comes from the face emission while the methane accumulation at the upper TG corner is caused by goaf methane emission;
- Irrespective of the cutting sequence, the methane concentration at the TG drum is higher than it at the MG drum, and is the highest along the face;
- The monitored methane concentration at the tail end of the shearer body can be approximately two or three times lower than the maximum concentration around the TG drum;
- Irrespective to the cutting sequence, the methane concentration is relatively low along the first half of the face, and its accumulation gradually shows up in the second half of the face;
- Methane emitted from the goaf is likely to accumulate at the middle and TG side of the goaf, and its distribution is significantly affected by the shearer’s position.

To obtain a better understanding of the methane flow characteristics under various operating conditions, a series of parametric studies were carried out, based on which the following conclusions can be summarised:

- The methane concentration around the shearer and at the inner TG corner is very sensitive to the variation of face flow rate, it decreases significantly as
the increase of flow rate, however, at the upper TG corner, the methane concentration does not drop significantly even when the flow rate is increased by 20 m$^3$/s;

- When the methane emission rate is increased along the face, the methane concentration will increase greatly if the face flow rate is not adjusted accordingly;

- The methane distribution at the immediate goaf and at the upper TG corner of the face is significantly affected by the increase of methane emission from adjacent gas bearing strata, and the methane level at the upper TG corner may become the highest along the face depending on the emission rate increase;

- The use of a TG cut through behind the face has significant influence on the methane level at the upper TG corner. This method transforms the “U” ventilation scheme to the “U+L” ventilation, which is much effective in diluting the methane accumulated at the upper TG corner;

- Methane level in the vicinity of drums and downstream is reduced greatly with the operation of drum sprays, and there is no significant rising tendency of methane concentration, indicating the dispersion effect of drum sprays on the methane accumulation around the drums;

- The impact of a MG curtain on the methane distribution is confined only to the first third of the face, where the methane diffusion is restricted to some extent, and its impact is negligible further downstream of the face.
6 MODELLING OF GAS FLOW CHARACTERISTICS IN A SINGLE LONGWAL GOAF

6.1 Introduction

Ventilation management on the longwall face is essential for methane dilution along the face. Whilst the ingress of ventilation airflow into the goaf also has a significant influence on the spontaneous heating of coal in the goaf, if not managed properly, it can potentially develop into a fire disaster. Unlike the longwall face, the goaf is formed due to the caving of roof rocks which determine its inaccessible characteristics, and there is no effective method to monitor the evolution of goaf gas except the tube bundle systems that can be used to monitor gas composition along the perimeter of the goaf. With the use of CFD modelling technique, it is possible to predict gas composition changes in the concealed caved goaf areas; together with the advanced data visualisation technique, it is possible to examine the unseen areas inside the goaf, and the evolution of goaf gases, especially oxygen, which determines the distribution of spontaneous combustion (sponcom) zones, can be better illustrated in three dimensional views. This is of particularly significant in terms of early prediction and location of goaf heatings, allowing the deployment of proactive prevention strategies prior to mining rather than testing various control measures during mining which could cause safety issue and affect production performance.

This chapter describes the application of the CFD modelling technique to investigate the goaf gas distribution in an active goaf using Y type ventilation, with special attention to the prediction of oxidation zones behind the face and the optimisation of goaf inertisation strategies.

The study was conducted based on the condition of Fenghuangshan Mine, a Chinese underground coal mine known for its high quality coal. As the coal produced by the mine is high quality anthracite coal, there is always a demand to maximise the coal extraction rate, and the most common approach to achieve this would be using the non-pillar mining methods. Although the coal is identified as spontaneous combustion prone, the non-pillar mining methods was adopted for its 154307
longwall face to minimise the number of coal pillars and simultaneously increase the coal extraction rate, forming the Y type ventilation to the face.

6.1.1 Geological conditions of 154307 longwall face

The 154307 longwall face was extracting 15# coal seam with 0.29 m of embedded rock in the middle of the seam. From the floor to the roof, it is 0.64 m coal, 0.29 m embedded rock and 1.22 m coal. The dip of the coal seam varies from 0° to 11°, and the average dip is 6°, the density of coal is 1.5 t/m³. The general occurrence of the coal seam is stable. The longwall face is 176.4 m wide and is 783.4 m in the strike direction of the coal seam. The coal seam thickness varies from 1.63 m to 3 m, and the average mining height is 2.2 m. The roof and floor conditions are shown in Table 6.1.

Table 6.1 The roof and floor conditions of 154307 face

<table>
<thead>
<tr>
<th>Roof type</th>
<th>Lithology</th>
<th>Thickness/m</th>
<th>Lithologic characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>The main roof</td>
<td>K₂ limestone</td>
<td>8.26-12.01</td>
<td>Dark gray, dense and hard, contain animal fossil and flint</td>
</tr>
<tr>
<td></td>
<td></td>
<td>9.94 on average</td>
<td></td>
</tr>
<tr>
<td>The immediate roof</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>mudstone</td>
<td></td>
<td>0.8-3.16</td>
<td>Black, contain plant fossil</td>
</tr>
<tr>
<td></td>
<td></td>
<td>1.77 on average</td>
<td></td>
</tr>
<tr>
<td>The main floor</td>
<td>Aluminum mudstone</td>
<td>3.72-9.65</td>
<td>Light gray - dark gray, contain pyrite</td>
</tr>
<tr>
<td></td>
<td></td>
<td>7.08 on average</td>
<td></td>
</tr>
</tbody>
</table>

6.1.2 Ventilation system of 154307 longwall face

Figure 6.1 shows the layout and ventilation system of the 154307 longwall face, in which both the belt road and the rail transport road are used to bring fresh air to the face. As the face retreats from the start-up line, caved goaf is formed immediately behind the face, and the rail (track) roadway behind the face (which would collapse normally) is maintained by building a retaining wall close to the longwall goaf, and is used as a bleeder road for air return, thus forming the Y type ventilation.
A volume of 755 m\(^3/min\) of ventilation air was supplied to the longwall face, of which 555 m\(^3/min\) was brought to the face through the belt roadway and 200 m\(^3/min\) through the track roadway. However, the actual flow rate to the face was slightly larger than that designed. Table 6.2 shows the actual ventilation variation during the face retreat.

Figure 6.1 Longwall face layout and ventilation system

Table 6.2 The actual ventilation rate to the 154307 face

<table>
<thead>
<tr>
<th>Date</th>
<th>Belt roadway (m(^3/min))</th>
<th>Track roadway (m(^3/min))</th>
<th>Return (m(^3/min))</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.11</td>
<td>980</td>
<td>220</td>
<td>1340</td>
<td></td>
</tr>
<tr>
<td>4.18</td>
<td>941</td>
<td>316</td>
<td>1312</td>
<td></td>
</tr>
<tr>
<td>5.13</td>
<td>530</td>
<td>222</td>
<td>793</td>
<td>Stopped</td>
</tr>
<tr>
<td>5.26</td>
<td>560</td>
<td>222</td>
<td>821</td>
<td>Stopped</td>
</tr>
<tr>
<td>7.1</td>
<td>784</td>
<td>264</td>
<td>1148</td>
<td></td>
</tr>
</tbody>
</table>

6.2 Goaf gas distribution - field monitoring data analysis

6.2.1 Field monitoring scheme

The longwall face is located in the 15# coal seam which has been evaluated as a sponcom liable seam. Due to the use of Y type ventilation scheme, significant air leakage could be induced into the goaf, the sponcom risk is greatly increased, therefore, it is necessary to monitor the gas composition along the retaining roadway side of the goaf, especially the carbon monoxide (CO) level which is known as the indicator gas of sponcom. When the longwall face had advanced to the 848 m position, 26 monitoring points had been set up, the specific locations relative to the longwall face are shown in Table 6.3 and Figure 6.2. Gas samples from each monitoring point were taken every two days and were analysed using the Gas Chromatography (GC), through which nine kinds of gases could be identified, CO,
CO₂, O₂, CH₄, C₂H₆, C₂H₄, C₂H₂, H₂ and N₂. The layout of each monitoring point is illustrated in Figure 6.3.

Figure 6.2 Layout of goaf gas monitoring points
### Table 6.3 Location of monitoring points relative to longwall face start-up

<table>
<thead>
<tr>
<th>No.</th>
<th>1#</th>
<th>2#</th>
<th>3#</th>
<th>4#</th>
<th>5#</th>
<th>6#</th>
<th>7#</th>
<th>8#</th>
<th>9#</th>
<th>10#</th>
<th>11#</th>
<th>12#</th>
<th>13#</th>
</tr>
</thead>
<tbody>
<tr>
<td>Location (m)</td>
<td>26</td>
<td>43</td>
<td>72</td>
<td>101</td>
<td>132</td>
<td>159</td>
<td>187</td>
<td>217</td>
<td>235</td>
<td>287</td>
<td>310</td>
<td>343</td>
<td>372</td>
</tr>
<tr>
<td>No.</td>
<td>14#</td>
<td>15#</td>
<td>16#</td>
<td>17#</td>
<td>18#</td>
<td>19#</td>
<td>20#</td>
<td>21#</td>
<td>22#</td>
<td>23#</td>
<td>24#</td>
<td>25#</td>
<td>26#</td>
</tr>
<tr>
<td>Location (m)</td>
<td>401</td>
<td>435</td>
<td>464</td>
<td>495</td>
<td>529</td>
<td>560</td>
<td>591</td>
<td>620</td>
<td>668</td>
<td>699</td>
<td>731</td>
<td>760</td>
<td>790</td>
</tr>
</tbody>
</table>

![Figure 6.3 Setup of a typical monitoring point](image)

#### 6.2.2 Field data analysis

Gas samples obtained from the underground monitoring points were analysed in the lab using the GC to investigate the dynamic variations of goaf gas as the longwall face retreated.

Figure 6.4 shows the goaf gas distribution along the goaf when the longwall face had retreated 210 m. It can be seen that the oxygen concentration gradually decreases from the face to the start-up line of the goaf, reaching 17% at 1# monitoring point. The sum of CO and C₂H₆ were selected to evaluate the degree of spontaneous heating in the goaf. As can be seen from Figure 6.4, a certain amount of CO and C₂H₆ had been detected in the goaf. Generally, the total concentration of CO and C₂H₆ varies significantly along the goaf, with the peak value of 19 ppm appearing in the middle of goaf.
Figure 6.4 Goaf gas distribution when longwall face retreated 210 m

Figure 6.5 shows the goaf gas distribution along the goaf when the longwall face had retreated 506 m, where the same decreasing trend for oxygen distribution can be found. The oxygen concentration was stable at around 20% within 380 m behind the face, and the lowest oxygen concentration was detected at 1# monitoring point (still above 16%), indicating significant fresh air ingestion into the goaf through the retaining concrete wall. While the total concentration of CO and C_2H_6 shows a steady increase from the start-up line towards the face, and it reached the maximum of 17 ppm immediately behind the longwall face, implying a great possibility of spontaneous heating in the goaf not far from the face.

Figure 6.5 Goaf gas distribution when longwall face retreated 506 m

When the longwall face retreated 848 m from the start-up line, the gas distribution in the goaf is shown in Figure 6.6. It can be seen that the general variation of oxygen
concentration follows the same decreasing trend, and it drops to 14% within 120 m away from the start-up line, slightly drop is also observed in the following 300 m when compared with Figure 6.5. Small fluctuations of CO and C₂H₆ can be seen 100 m behind the face in the goaf, with an average value of 10 ppm, however, the gaseous concentration dramatically increased to 16 ppm immediately behind the face.

Figure 6.6 Goaf gas distribution when longwall face retreated 848 m

Therefore, it can be concluded from the field monitoring data that:

- The oxygen concentration is generally high in the goaf along the retaining wall side, during the retreating of longwall face, only within 120 m away from the start-up line that the oxygen concentration was lower than 20% (but still higher than 14%), all the other sampling points along the retaining wall side come with an average of 20% oxygen concentration, indicating severe fresh air leakage into the goaf either from the face or from the retaining wall;

- At the start up stage of longwall face, the maximum concentration of CO and C₂H₆ appeared in the middle of goaf, while with the retreat of longwall face to a further distance, the peak value gradually migrated to an area of 100-200 m behind the face, which suggested the oxidation of coal mainly occurred within a certain distance behind the face, and the corresponding spncom gases were carried to the return flow stream by the leaked air to the goaf;

- With the retreat of longwall face, the total concentration of CO and C₂H₆ gradually became stable, and it was slightly lower compared with that in the
goaf immediately behind face, this is due to the continuous dilution of increased air ingress from the retaining wall into the goaf.

6.3 Development of CFD models

A three dimensional CFD model was developed to represent the longwall face which was 780 m in length, 176.4 m in width and 80 m in height to cover the caved and fractured zones in the goaf. Figure 6.7 shows the geometry of the CFD model, boundary conditions and the computational grid. It can be seen from Figure 6.7 that the boundary conditions involved in the model are the velocity inlet at the two entries (belt road and rail road), and the pressure outlet at the only exit of the computational model. Other boundaries of the model, such as the roadway roof, floor, ribs and the goaf boundaries were treated as standard walls. The longwall goaf is treated as a porous media, and flow through the porous regions was modelled by the addition of a momentum source term to the standard fluid flow equations. Instead of constructing an actual wall in the physical model which seals the goaf completely, the retaining wall was modelled with a low permeability zone using the UDF function to allow certain volume of air and gas flow leakage to the return airflow.

(a) Plan view of the CFD model geometry
Figure 6.7 Longwall CFD model geometry and computational grid

6.4 Base model validation and results

As an integrated part of the CFD numerical modelling, validation of base model results was carried out using the field gas monitoring data. Figure 6.8 shows a comparison of the predicted oxygen concentration levels and field monitoring data inside the retaining wall along the goaf. The CFD modelling results show that the oxygen ingress inside the retaining wall remains as high as 20% even up to 700 m behind the longwall, indicating that significant air leakage between the retaining wall and the boundary of consolidated goaf. A good agreement can be observed between
the base CFD modelling result and field gas monitoring data. The base model was then used to investigate the spatial distribution of spontaneous heating zones and proactive goaf inertisation strategies.

![Figure 6.8 Base CFD model validation - model result vs. field monitoring data](image)

Figure 6.9 shows the CFD predicted oxygen distribution and the spatial distribution of spontaneous combustion zones in the goaf. CFD results indicate that fresh air from the belt road (inlet 1) leaks into the goaf along the rib side to a depth up to 300 m, the leaked air partly travels further towards the face start-up line, and partly penetrates across the unconsolidated goaf, but mostly turns back along the unconsolidated goaf edge to join the main air stream behind the chocks within 50 m in the goaf; it then merges with air leakage from the face and rail road (inlet 2), and travels along the goaf boundary strip inside the retaining wall until the start-up line, before reporting to the return roadway. Figure 6.9(b) shows the 3D Iso-Surfaces of oxygen concentration at 7% and 18%, between which is the oxidation zone where the heating of residual coal is mostly to occur. Figure 6.10 and Figure 6.11 respectively show the oxygen concentration distribution at 50 m, 100 m, 200 m, 300 m and 400 m (cross the goaf) behind the face and at different roof levels (0 m, 20 m, 40 m and 60 m above the coal seam roof). As can be observed from the plot, this area is spatially distributed in the goaf on the belt road side up to 200 m, 50 m behind the face, and along the retaining wall side about 60 m into the goaf. Beyond this area are the cooling zones (oxygen level is above 18%) where excessive air leakage will dissipate any oxidation heat to support the self-heating process, and the choking zone (oxygen
level is below 7%) where the oxygen level would be insufficient for coal to undergo active spontaneous heating.

(a) Plan view of oxygen distribution in the goaf

(b) 3D view of oxygen distribution in the goaf
(c) Spatial distribution of potential spontaneous heating zones

Figure 6.9 Oxygen distribution and spatial distribution of spontaneous heating zones in the goaf

Figure 6.10 Oxygen concentration distribution on roof level at different distances behind face (vertical to the face retreating direction)
Figure 6.11 Oxygen concentration distribution on different roof level in the middle of face (from face to the start up line)

CFD model results show that the use of Y type ventilation can induce serious air leakage spatially into the goaf with a wide range of oxidation zones that are favourable to the development of spontaneous heating. The most likely areas for spontaneous combustion to occur in the goaf (at mining level) are the goaf edge on the belt road side some 200 m behind the face and 60 m into the goaf; within 50 m behind the chocks, and the unconsolidated areas some 60 m inside the retaining wall along the bleeder (return) road till the start-up line. Goaf gas composition along the retaining wall must be carefully monitored to detect the onset of any heating and avoid delayed control actions against the occurrence of a possible heating in these areas. Obviously any proactive measures such as goaf inertisation should be targeted towards these areas.

6.5 Parametric study of goaf inertisation strategies

An effective method to prevent the development of self-heating in longwall goafs is to implement proactive inertisation by pumping inert gas such as nitrogen or carbon dioxide into the goaf to minimise the area of potential oxidation zones. To develop the optimum inertisation strategies, a set of parametric studies were conducted using the CFD model to identify the optimum injection point behind the longwall face. In
addition, the impact of ventilation adjustment was also investigated using the CFD model.

6.5.1 Inertisation from belt road side

Figure 6.12 shows a comparison of oxygen distribution in the goaf at the mining level after inert gas (pure nitrogen) injection at 30 m, 50 m, 100 m and 200 m behind the face on the belt road side at a rate of 0.25 m$^3$/s. It can be seen that the oxidation zones (refer to Figure 6.9(a)) on the belt road side and behind the chocks has been eliminated by injecting inert gas at more than 50 m behind the face while the inertisation has limited effect inside the retaining wall side, unless the injection is conducted at some 200 m behind the face. It also indicates that the inert gas injection point should not be too close (i.e., less than 50 m) to the face where the goaf is not completely compacted and air leakage is relatively high, because the inert gas will be easily dispersed by air leakage as soon as it is injected.

(a) Inert gas injection 30 m behind face

(b) Inert gas injection 50 m behind face
6.5.2 Inertisation from retaining wall

Figure 6.13 shows the effect of goaf inertisation when injection is carried out on the retaining wall side. CFD modelling results show that injection on the retaining wall side only has a limited effect on areas around the injection point, as much of the inert gas will be flushed away by air leakage inside the retaining wall. Again, this demonstrates that good inertisation effect cannot be obtained if injection is carried out at high air leakage areas. There is almost no inertisation effect in oxidation zones on the belt road side and immediately behind the chocks. Consequently, goaf inertisation should not be conducted on the retraining wall side, unless it is needed to deal with localised heating in combination with other control measures such as multiple injection points (Figure 6.13(c) and (d)), foaming or temporary stoppings to minimise air leakage into these areas.
Inert gas injection 50 m behind face

Inert gas injection 100 m behind face

Inert gas injection simultaneously 50, 100 and 200 m behind face

Inert gas injection simultaneously 100, 200 and 300 m behind face

Figure 6.13 Goaf inertisation on oxygen distribution - injection from retaining wall
6.5.3 Inertisation behind chocks

Figure 6.14 shows the effect of goaf inertisation when injection is carried out behind the chocks. It can be seen that relatively good goaf inertisation effect can be achieved except injecting inert gas at 30 m behind face and 20 m away from the retaining wall (Figure 6.14(a)), the majority of inert gas injected is able to stay around the injection point and gradually migrates towards the boundary of the goaf, converting part of the oxidation zones behind the face to an inert environment.

The model results indicate that injecting inert gas behind chocks in the goaf is a relatively good inertisation strategy, but the injecting point (end of the injection pipe) should be located close to the belt road side at least 50 m behind the face, which is a better place for the inert gas to migrate towards the oxidation zones along the perimeter of the goaf, thus reducing the spatial distribution of oxidation zone and lowering the risk of backflow of inert gas to the working face.

(a) Inert gas injection 30 m behind face and 20 m away from the retaining wall

(b) Inert gas injection 30 m behind face and 40 m away from the retaining wall
6.5.4 Inertisation using surface goaf wells or boreholes

Under certain circumstances when underground access becomes not possible (e.g. mine evacuation), inertisation can be conducted using existing wells or boreholes which are usually used for gas drainage. In this case, it is assumed that the surface wells are located 60 m away from the belt road side at an interval of 200 m, and the first well is 130 m behind the face. Figure 6.15 shows the goaf inertisation effect of inert gas injection from a single surface well/borehole and a combination of two or
four goaf wells. It can be seen that the inertisation of high oxygen areas in the goaf can be more effectively achieved by injecting inert gas via surface goaf wells/boreholes. Modelling results show that a combination of surface wells should be used to inject inert gas for goaf inertisation when surface access can be obtained, rather than delivering inert gas underground. At least, the practice of using a single borehole for inertisation at about 100 m following the face, as shown in Figure 6.15a, should be adopted for longwall systems using Y ventilation (or bleeder road).

(a) Inert gas injection from the first surface well

(b) Inert gas injection from the second surface well

(c) Inert gas injection from the first and third surface wells
6.5.5 Impact of ventilation adjustment on goaf gas distribution

From the point of field implementation, besides improving the mining technology (increase face advancing rate and reduce the amount of residual coal in the goaf), strategies that could be used for sponcom control include optimising ventilation, reducing ventilation resistance of the face and enhancing the sealing of goaf. It is also recommended that air curtains or brattices should be used at the intersections between air entries and the face. Regarding the Y type ventilation system developed at 154307 face, it is vital to keep the integrity of the retaining wall to maintain the effective ventilation section and reduce the ventilation resistance of the retaining roadway.

When the ventilation provided to the face through the belt roadway and track roadway are reduced to 600 m$^3$/min and 150 m$^3$/min respectively, both the oxygen and nitrogen concentration distribution in the goaf is illustrated in Figure 6.16. It can be seen that the range of oxidation zones on the belt roadway side and immediately behind the face are reduced to some extent compared with that of the base model (refer to Figure 6.9).
6.6 Summary

This chapter discussed the use of three dimensional CFD models to investigate the distribution of spontaneous combustion zones for a Chinese longwall face using Y type ventilation. Firstly, to understand the gas distribution in the goaf, field monitoring was conducted along the retaining wall of the goaf using a tube bundle system. During the retreat of a longwall face, a relatively higher and stable concentration of sponcom gaseous products (CO and C₂H₆) were detected immediately behind the face, and they gradually dropped towards the outlet, indicating a continuous self heating of coal was occurring within a certain distance behind the face as the face retreated. In addition, the area along the retaining wall in the goaf was identified as oxygen rich due to severe air leakage through the retaining wall. Then the monitored oxygen concentration was used to validate the base CFD model, and good agreement with the two sets of data was demonstrated. Modelling results show that the most likely areas (7~18% oxygen) for spontaneous combustion to occur spatially in the goaf are the goaf edge on the belt road side some 200 m
behind the face and 60 m into the goaf; within 50 m behind the chocks, and in the unconsolidated areas about 60 m inside the retaining wall along the bleeder (return) road till the start-up line. Air leakage is most excessive in the unconsolidated goaf boundary inside the retaining wall along the bleeder road.

Based on above understandings, it is recommended that both goaf gas monitoring and proactive inertisation should be conducted to minimise the occurrence of spontaneous goaf heating. Through a series of parametric studies, it was demonstrated that an optimum goaf inertisation can be achieved by pumping inert gas (nitrogen at a rate of 0.25 m$^3$/s) at least 100 m behind the face on the belt road side, or ideally, if surface access permits, via surface goaf hole(s). Goaf inertisation on the retreating wall side will only be effective for localised heating, and should be used in combination with other control measures to minimise air dilution in these areas.
7 MODELLING OF GAS FLOW CHARACTERISTICS IN MULTI-GOAF AREA

7.1 Introduction

As a longwall face retreats, goaf is formed behind the face. Once a longwall face retreats to the finish-off line, longwall equipment is transferred to the following longwall face, and the goaf formed behind the old face will be completely sealed, this goaf can be referred to as sealed goaf in comparison with the active goaf formed behind a retreating longwall face. As mining progressively transfers to the newly formed adjacent longwall face, a large mined out area with several sealed goafs will be formed. Generally, these sealed goafs are isolated by chain pillars between old faces; however, with the extension of this mined out area, the chain pillars may collapse to some extent, and the seals in the cut throughs between adjacent goafs may be partially or completely destroyed, allowing gaseous exchange between the goafs. Under certain circumstance, a perimeter roadway around the sealed goafs and an active goaf is ventilated to flush the hazardous gases away from the active longwall face. One drawback of this ventilation system is that fresh air may be able to penetrate into the sealed goafs through the previously constructed seals, increasing the risk of spontaneous heatings in both the sealed and active goafs, and the situation would be worse with the occurrence of explosive gases. Therefore, it is vital to employ tube bundle systems to monitor the goaf gas distribution along the goaf boundaries. Meanwhile, the integrity of seals in the cut throughs connected to the goaf must be checked regularly.

It is known that the use of a tube bundle system is limited to the boundaries of the goaf due to the caving characteristics of the goaf and it cannot be used to evaluate the flow patterns of goaf gases. As mentioned in Chapter 6, CFD modelling technique is capable of predicting the distribution and migration of goaf gases within the goaf, in particular oxygen, which is vital to determine the distribution of sponcom prone zones for early prediction and location of potential heatings. Thus, CFD studies were conducted in this chapter to provide an improved understanding of the goaf gas flow
patterns in general and the behaviour of different gases (CH\textsubscript{4}, CO and C\textsubscript{2}H\textsubscript{6}) under the influence of different operational changes.

The main contents of this chapter involve a critical review of available data and information, both from field monitoring and documented reports for the development of 3D Computational Fluid Dynamics (CFD) models to help understand the fundamental characteristics of goaf gas flow patterns in the active (goaf behind the working face) and adjacent sealed off goafs. Specifically it will include:

- Critical review and examination of the current ventilation, goaf monitoring and gas drainage data, as well as longwall panel layout and geological settings of longwalls;
- Development of 3D CFD models to include the active and sealed off goaf areas to model the goaf gas flow and goaf heating gaseous products (mainly CO and ethane) and dispersion patterns under current working conditions;
- Validation of the above CFD models by using field data (both tube bundle data and bag sample data);
- Parametric studies by using the validated CFD model to investigate the behaviour and flow pattern characteristics of goaf gas in longwall goafs, and the impact of various operational parameters. Specific scenarios include:
  - active goaf upon sealing, with regulation applied in new LW TG to draw goaf atmosphere back towards seals;
  - flood ventilation of flank/perimeter road;
  - sealing flank roadway, and its effect upon sealed goaf;
  - goaf inertisation using nitrogen and/or methane from underground inseam gas drainage systems.
7.2 General information of the mine

7.2.1 Mandalong mine

Centennial Mandalong mine is located 50 km south of Newcastle, NSW. The mine operates a retreat longwall panel about 150 m wide in the West Wallarah Seam of the Newcastle Coalfield. The landforms vary from flat floodplain to the foothills of the Watagan Mountain range, with overburden cover varying from 160~350 m and vertical difference in seam 40~60 m between lowest to highest points. Figure 7.1 shows the mine surface topography and overburden in relation to the longwall mine layout.

Figure 7.1 Longwall layout and topography

Figure 7.2 shows the typical stratigraphic section and working seam cross-section of the mine site. The seam varies in thickness from 3.6~6.5 m and has moderate gas content up to 6 m$^3$/t. The Fassifern seam underlies the West Wallarah seam, with the interburden thickness varying between 4~8 m. The coal seams dip down initially from the longwall boundary inbye then rises up as the longwall retreats towards the Mains. The predominant seam gas constituent is methane, but ethane is also present in appreciable amounts as a subordinate component. Permeability ranges from
2~2000 mDarcy, average 5~15 mDarcy. During longwall retreat methane from the
Fassifern is liberated into the active goafs. Mandalong mine operates a gas drainage
plant at 450~1450 l/s for in seam drainage to reduce rib emissions.

Figure 7.2 Typical stratigraphic and cross-section of Mandalong Mine
Mandalong colliery consists of two major parts: Mandalong longwall panels and Cooranbong mine workings. The mine is served with seven heading Mains with two heading LW gateroads, as shown in Figure 7.3. Access to the mine is via a men and material drift. Mandalong Mine is ventilated by two fans which are capable of producing 450 m$^3$/s at 4.5 kPa and Cooranbong mine workings are ventilated by a single Cooranbong fan (2 fans installed). Cooranbong and Mandalong ventilation circuits are segregated at 15c/t Mandalong Headings.

![Mandalong LW workings](image)

**Figure 7.3 Longwall panel layout at Mandalong Mine**

Longwall panels are ventilated by the simple “U-type” system with a Perimeter Roadway surrounding the goaf by seals. Goaf gas monitoring instruments are installed at regular intervals along the perimeter roadway to allow the inspection and maintenance of seals. Gas monitoring systems in place include real-time Safegas tube bundle, Gas chromatograph Varian CP-4900 and a Bag sampling regime which is typically every 2nd seal every week, except TG1 every 4th seal every week (30 m centres).
7.2.2 Longwall goaf geometry

At the time of this study, Mandalong mine operated LW9 of 155 m width with active goaf forming behind as the face retreated. The sealed goaf areas include LW1-LW8 with abandoned pillars between the mined longwalls. Longwall panel widths were set at 120-160 m, the length of the longwall panels are between 2020 and 3020 m. Each panel has two headgates for air intake and cut throughs connecting the two maingates developed at an interval of about 100 m. The first maingate which had served as intake during LW8 retreat now serves as a tailgate for air return of LW9. The width of the chain pillars in the sealed goaf is around 46 m rib to rib. The dimensions of roadways for ventilation and cut throughs in sealed goaf are typically of 5.2 m×3.4 m. More detailed profile and dimensions of the longwall goaf are shown in Figure 7.4.

Figure 7.4 Longwall goaf geometry
Figure 7.5 shows the floor contour of the West Wallarah seam. It can be seen that the elevation of the caved goaf areas in the central parts can have significant changes as the longwalls retreat in the coal seam. From LW1 to LW8, the vertical elevation of goaf drops deeper, and in the retreat direction, the elevation rises for LW1 to LW6 but dips slightly near the start-up of longwall and then increases for LW7-LW9. For most of the sealed goafs, the elevation changes greatly in the middle part of the panels.

7.2.3 Longwall ventilation

As described above, Mandalong mine consists of two major underground parts: the Mandalong longwalls and the mined Cooranbong workings. Two main fans are used for ventilating the whole region. The fan in Cooranbong operates at 100 m$^3$/s and the
other one in Mandalong is running at 250–350 m$^3$/s. As the Cooranbong part has been mined and sealed already, this study only focuses on longwall goafs at Mandalong mine.

Standard U-type ventilation has been used to ventilate all longwalls. Between two adjacent panels, high integrity seals were installed in cut throughs to avoid air especially oxygen going into the mined area. The perimeter roadway is maintained around the goafs and used to allow the inspection and maintenance of seals and the start-up of longwalls. With the extraction of the working face, the ventilation rates in the working face and perimeter roadway are adjusted depending on the coal mine management and the gas flow patterns in the goaf. Figure 7.6 shows a typical ventilation system and air flow rates to the face and along the perimeter road.

Figure 7.6 A typical ventilation system in Mandalong Mine
7.2.4 Gas reservoir characteristics and gas emission monitoring

There is significant data describing the gas reservoir characteristics for Mandalong mine. The models used to predict specific emissions range significantly depending on the amount of gas emission from the strata underlying the West Wallarah workings section. Experience gained from the extraction of LW1-2 indicated that gas make reporting to the return appeared to be contributed from the coal being mined and the remaining coal left in the immediate floor. Field data from the extraction of Longwalls 3-4 indicated that gas emissions from underlying strata also made an appreciable addition to the gas make from the face or goaf. The gas content of West Wallarah seam at various locations is shown in Figure 7.7.

![Figure 7.7 West Wallarah seam gas content (m³/t)](image)

Longwall cutting sections are located in the upper 3.5 to 5.0 m of the West Wallarah seam. As the seam varies in thickness from 3.5 to 7.0 m in the proposed longwall blocks, a section of the floor coal will remain in the goaf, from almost nothing up to about 2.0 m. The Fassifern seam varies from 1.5 to 3.5 m thick and lies below the
West Wallarah seam at close distances ranging from 5 to 8 m. The spatial relationship between longwall workings and the Fassifern seam is illustrated in Figure 7.8. It is expected that gas liberated from the Fassifern seam due to longwall de-stressing will migrate into the longwall goafs in the West Wallarah seam.

Figure 7.8 Generic longwall excavation layout in relation to Fassifern seam

Gas emission has been in the order of 850 l/s towards the inbye end of LW4 with peak rates of around 1200 l/s also experienced. The outbye end of LW3 experienced gas emissions in the order of 200 l/s and LW5-6 experienced gas flows up to 1350 l/s in the inbye half of the longwall blocks. In addition to methane, ethane has also been detected at 30 ppm in older areas and up to 225 ppm in more recent goaf areas, which could be partly generated from slow oxidation of residual coals left in the goaf areas.

As mentioned above, Mandalong mine mainly relies on a tube bundle system and bag sampling practices behind goaf seals to monitor the changes of goaf gas compositions. Figure 7.9 provides the snapshots of goaf gas distribution patterns at different stages in the active goaf following longwall retreat and along the edges of the sealed areas. An examination of the gas monitoring data along the goaf edge of the longwall panels showed that:

Face at start-up stage:

- Oxygen penetration into the active goaf (LW9) is high, up to 18%, and methane around 14%;
- Oxygen level remains higher at all seals in the back of the goaf areas, up to 15% behind LW1, then drops down to around 1~2% in seals along LW1 and along the Mains;
• Methane level remains around 30% at most seals with the highest value up to 67% recorded at seals of LW7 and LW8 close to the Mains.
• High oxygen areas are in the active goaf, and around the seals at the back of the longwall blocks.

As face progresses to middle stage:

• Oxygen penetration into the active goaf remains high, up to 18% even some 600 m behind the face, and 14% beyond 1000 m; and methane concentration gradually starts to build up to 27%;
• Oxygen levels at the seals in the back of the goaf areas start to drop down slightly, but still remains around 10% behind LW1, 2% around seals along the perimeter road, and slightly up along the Mains close to 4%;
• Methane level starts to build up in the goaf areas, above 40% at most seals with the highest value up to 57% recorded at seals of LW7 and LW8 close to the Mains.

As face approaches to finishing off stage:

• Oxygen penetration into the active goaf still remains high, up to 18% even some 900 m behind the face, and 12.6% near the start-off line, with methane concentration up to 30%;
• Oxygen levels continue to drop around all the seals in the goaf areas, mostly well below 5%, but still remains above 10% behind LW1;
• Methane buildup appears to be more evenly distributed in the goaf, at about 40~50% around most seals with the highest value up to 57.9% recorded at seals of LW7 and LW8 close to the Mains.

High level of oxygen ingress into the goaf could lead to the development of spontaneous heating behind the longwall face, particularly during face stoppage or low advance rate of the panel. Gas monitoring results indicate that the most likely areas prone to the onset of spontaneous heating in the goaf would be:
- The active goaf, particularly along the perimeter road (MG side), up to 1000m behind the face;
- Face start-off areas, particularly for LW8, LW9 and LW1.

The goaf gas monitoring data is also used to calibrate the base CFD models which are then used for parametric studies in this project.

(a) Goaf gas composition at various locations - start-up stage
(b) Goaf gas composition at various locations - middle stage
7.3 Development of CFD longwall models

As described above, Mandalong mine has a large goaf area connected to the active longwall by a perimeter roadway and sealed cut-throughs and abandoned chain pillars. To understand the behaviour of goaf gas flow in such a large goaf area, detailed longwall geometry data and seam floor contours are used to build the geometrical model of the longwall goaf. Figure 7.10 to Figure 7.12 respectively show the CFD model geometry and computational grid. Due to the complexity of the
geometry, the model was meshed using a combination of hexahedron and tetrahedron mesh with over 1.5 million computational grids.

A major challenge in developing the CFD model has been the changing goaf elevations along the different parts of these longwall blocks. To take into account floor elevation of caved goaf, three base CFD models respectively portraying longwall at start-up, middle and finish-off position were built, each consisting of eight sealed goafs and an active goaf behind LW 9 panel. Specific geometry of the longwall panels is in line with Figure 7.6. The height of the longwall model is 80 m which includes 10 m of floor strata (to include the Fassifern seam), 4m average cutting height, and 66 m above the mined seam roof. A User Defined Function (UDF) has been coded to represent the spatial permeability distribution and gas emissions in the goaf and hooked to the main CFD solver. The UDF is also used to define all the seals around the goaf areas.

Figure 7.10 CFD model geometry for LW9 retreating at different stages - plane view
Figure 7.11 CFD model geometry - 3D view
7.4 Base CFD model results

Three base CFD models were developed to simulate goaf gas flow behaviour when LW9 retreats at the start-up, middle and near finish-off positions.

7.4.1 Base Model 1 - Operating longwall (LW9) at start-up stage

This CFD model was built to investigate the goaf gas flow behaviour when the new longwall block (LW9) has retreated some 150 m from the start-up line. Figure 7.13 shows the ventilation system at the start-up stage based on which the base model 1 was set up. Figure 7.14 and Figure 7.15 respectively show the longwall goaf gas (methane and oxygen) distribution patterns at this stage.
Figure 7.13 Ventilation scheme at the start-up stage
Figure 7.14 Methane and oxygen distribution pattern in longwall goafs when LW9 at start-up stage (projection from roof)
(b) Cross-section view

Figure 7.15 Oxygen distribution pattern in longwall goafs when the longwall (LW9 is at the start-up stage)

The modelling results indicate that when the operating longwall (LW9) was at start-up stage:

- High oxygen ingress tends to appear at all the cut-throughs around the active longwall goaf, around the cut-throughs behind LW1-2, LW4-5, and the sealed corner goaf areas connecting the mains and LW9 TG;
- There is likely to be significant oxygen ingress via cut-throughs into the middle part of the sealed goafs of LW8 and LW7, mainly due to the lower elevations of the goaf and the buoyancy effect of methane leading to oxygen replacement of methane, a phenomenon has not been fully revealed by the tube bundle monitoring system due to lack of data in this area;
- High oxygen ingress will occur via cut-throughs around the start-off area of LW1;
- Methane gas tends to accumulate on the shallow edges of the goaf and gradually build up into the deep parts of the sealed areas towards finish-off lines of LW1-4, mainly due to ventilation pressure and the higher elevations of the goafs.

In general, modelling results indicate that the goaf gas flow behaviour is much affected by longwall ventilation pressure at the earlier stages, with oxygen penetration deep into active goafs, via cut-throughs around the perimeter road, and the middle part of the sealed adjacent longwall goafs.
7.4.2 Base Model 2 - Operating longwall (LW9) at middle stage

This CFD model was built to investigate the goaf gas flow behaviour when the operating longwall block (LW9) has retreated to the middle part of the panel. Figure 7.16 illustrates the ventilation system and Figure 7.17 shows the modelling results of the longwall goaf gas (methane and oxygen) distribution patterns at this stage.

![Ventilation scheme at the middle stage](image)

Figure 7.16 Ventilation scheme at the middle stage

The modelling results indicate that when the operating longwall (LW9) was approaching middle stage:
• Oxygen level remains high in the active longwall goaf with oxygen concentration still reaching 15% even at the 8th or 9th cut-through (900 m) behind the face. This is also evident in the field monitoring data;

• There is significant oxygen ingress from the active longwall into the adjacent part of the sealed goafs of LW8 and LW7 and this could extend further deeper into the goafs. This is mainly attributed to the lower elevations of the goaf and the buoyancy effect of methane leading to oxygen replacement of methane;

• Oxygen ingress still tends to occur around the cut-throughs along the perimeter road around the start-off line behind LW1-2, LW3-4, and the sealed corner goaf areas connecting the mains and LW9 TG;

• Oxygen concentration tends to drop slightly deep behind the face and along the edges of sealed goafs, as compared to the start-up position;

• Methane concentration starts to build up gradually in the active goaf (LW9) and the higher edges around the sealed goaf areas, with the highest methane accumulation in goaf areas close to the mains and the higher parts of the goaf close to the Perimeter Roadway.

In general, modelling results show that as the operating longwall moves towards the middle section of the block, face ventilation still exerts a significant impact on goaf gas flow behaviour, particularly in the active goaf and the middle parts of the adjacent sealed goafs; however this influence reduces gradually in deep goaf areas as the face moves away, thus resulting in methane build up in deep goaf inside the seals and at higher elevation areas. Modelling results show that there is significant oxygen ingress into the adjacent goafs as the operating face approaches the lower part of the West Wallarah seam. High methane concentrations above 50% start to build up in deep goaf areas and the higher parts of the goaf.
Figure 7.17 Longwall goaf gas (methane and oxygen) distribution pattern when the longwall (LW9) has retreated to the middle part of the panel

7.4.3 Base Model 3 - Operating longwall (LW9) at finish-off stage

This CFD model was built to investigate the goaf gas flow behaviour when LW9 has retreated close to the finish-off line of the longwall panel. The ventilation system at this stage is shown in Figure 7.18. Figure 7.19 and Figure 7.20 respectively show the modelling results of the longwall goaf gas (methane and oxygen) distribution patterns at this stage.
The modelling results indicate that when LW9 was at near finish-off stage:

- Oxygen levels remains high in the active longwall goaf with oxygen concentration reaching 15%, even at the 8th cut-through (800–900m) behind the face. However the extent of high oxygen ingress into adjacent sealed goafs has reduced and is now mainly limited to the active goaf area;
- Oxygen ingress into the sealed goaf areas tends to decrease significantly except cut-through seal areas around the Perimeter Roadway behind LW1-2.
- As the longwall has passed the lower parts of the adjacent sealed goafs in the middle section, ingressed oxygen has been gradually replaced by the build-up of methane in the goaf.
As the longwall approaches the finish-off line, methane build-up tends to be more uniformly distributed in the entire sealed goaf areas except in the goaf edge areas (near start-up lines) behind LW1-4 in the vicinity of the Perimeter Roadway. Methane concentration reaches above 50% in the majority of the sealed goaf.

In general, CFD modelling results show that as the operating longwall retreats to the final stage of the block, oxygen penetration into the active goaf still remains high up to 15% even some 800 m behind the face; however the extent of oxygen ingress into the sealed goaf has reduced significantly, allowing high concentration methane to build up in most part of the sealed areas.

Figure 7.19 Goaf gas (methane and oxygen) distribution pattern when the longwall (LW9) has retreated close to finish-off line (roof projection)
7.4.4 Summary of base model results

Results from the base models indicate that the overall goaf gas flow pattern changes as the operating longwall (e.g. LW9) retreats from the start-up position to finish-off line. This is mainly due to the changing ventilation pressure differential across the entire goaf areas as well as the varying goaf elevations along the longwall panels.

At the start-up stage, the impact of face ventilation is much more significant on the behaviour of goaf gas flow. High oxygen ingress occurs in the active goaf and around the seals along the perimeter road behind the sealed longwall panels, and high methane level tends to build up at those higher elevation areas close to the Perimeter Roadway and deep goaf areas towards the Mains. As the longwall retreats towards the middle part of the panel, oxygen ingress will increase significantly into the sealed goafs of adjacent longwalls (LW7-8), i.e., the lowest parts of the goaf area, and methane tends to build up at increased concentrations beyond this area, in particularly in zones close the Mains and the higher goaf edges long the Perimeter Roadway; When the longwall reaches the near finish-off line of the panel, the impact of face ventilation on goaf gas flushing in the sealed areas gradually diminishes and methane distribution in the goaf becomes more stable and regular throughout the sealed goafs. Oxygen ingress is largely confined to the active goaf and some areas around the start-up seals in LW1-4. In all cases, oxygen penetration into the active goaf remains high, reaching 15% or above even some 800 m behind the longwall face, and methane concentration comes up to above 50% in deep goaf areas.
The base CFD models have been adjusted against the limited field gas monitoring data and a reasonable agreement can be observed in the general pattern of goaf gas flow distribution in all cases. An interesting phenomenon revealed by CFD modelling is the likely extensive ingress of oxygen into the sealed goaf in the middle of the longwall blocks where the goaf dips down following the seam contours of West Wallarah seam. The CFD modelling results have provided further insights into the goaf gas flow patterns in the large sealed goaf areas where field monitoring has been impractical.

7.5 Parametric studies

On completing the above base model studies, a number of parametric studies have been carried out to investigate different scenarios relating to goaf management at Mandalong Mine. This section provides a summary of these parametric studies.

7.5.1 Ventilation impact on the gas flow patterns in the goaf area

Based on the base models, the impacts of face ventilation on goaf gas flow patterns at three different stages were studied by means of increasing the ventilation volume to the face by 15~20 m$^3$/s and simultaneously reducing the same quantity for the perimeter road without varying the total ventilation quantity.

Start-up stage:

Figure 7.21 shows the oxygen ingress patterns into the goaf areas when the face ventilation was increased by 15 m$^3$/s. In comparison with the Base Model 1, the modelling results show that oxygen ingress increased significantly in the active goaf, and the adjacent sealed goafs in the middle part of the longwall blocks, however the general pattern of the goaf gas flow distribution remains similar. Modelling results show that excessive ventilation at the start-off stage of the longwall could potentially increase oxygen penetration in both active and sealed off adjacent goafs and
consequently lead to goaf heating in these areas. Increased face ventilation at this stage is more effective for goaf gas flushing.

Figure 7.21 Oxygen distribution pattern in goaf with increased face ventilation - start-up stage

Mid-Stage:

Figure 7.22 shows the oxygen distribution pattern in the goaf area with face ventilation increased by 15m$^3$/s. It is noted that the general pattern of goaf gas distribution is similar to the base model; however, with the increase of face ventilation quantity, more fresh air tends to penetrate into both the active goaf and the middle part of sealed LW6-8 due to the combined action of pressure difference and buoyancy effect. Increased face ventilation at this stage is less effective for influencing goaf gas behaviour in other deep goaf areas inside the perimeter road.
Figure 7.22 Oxygen distribution pattern in goaf with increased face ventilation – mid-stage

**Finish-off Stage**

Figure 7.23 shows the oxygen distribution pattern in the goaf area when the face ventilation was increased by 15 m$^3$/s. It can be observed that increased face ventilation at this stage mainly leads to increased oxygen penetration in the active goaf and part of the sealed adjacent goafs at lower elevations. It has however, limited impact on the behaviour of goaf gas flow in other parts of the sealed goaf areas.

CFD modelling results indicate that increased face ventilation can have a significant impact on the behaviour of goaf gas flow patterns, particularly in active longwall goaf. High face ventilation is more effective for goaf gas flushing at start-up stages and is also likely to cause high oxygen ingress in the active goaf, adjacent goafs and around the seals along the perimeter road. As the face retreats, however, the impact of face ventilation on goaf gas behaviour gradually diminishes, and will mostly govern the distribution of goaf gas in the active goaf and part of the most adjacent sealed goaf.
Figure 7.23 Oxygen distribution pattern in goaf with increased face ventilation - finish-off stage

7.5.2 Impact of roof fall in the perimeter road

Base model 2 and model 3 were selected to investigate the gas flow patterns when a roof fall occurred in the perimeter road (as shown in Figure 7.24). The roof fall was modelled via the addition of high resistance plug in the perimeter road via the User Defined Function (UDF) code hooked to the solver. The ventilation quantity remained the same for the face while it was reduced by 10m$^3$/s for the perimeter road as a result of roof fall compared with the corresponding base model. Figure 7.25 shows the oxygen distribution under this situation for model 2 representing operating longwall retreated to middle stage of the longwall block.

Modelling results show that the quantity of air leakage into the goaf varies along the perimeter road behind the sealed goaf, and the closer the distance to the position where the roof fall occurs, the larger the air penetrating impact through the cut throughs, which also indicates the increased risk of goaf heating among these
locations. The results indicate that a roof fall or increased ventilation resistance due to roadway failure/collapse in the perimeter road will significantly increase air leakage via cut-throughs in the immediate vicinity of the fall and therefore increase the risk of goaf heating in these areas.

Figure 7.24 Roof fall point in the flank road

(a) Oxygen distribution in the goaf - plan view
Figure 7.25 Gas distribution pattern in goaf with roof fall in the perimeter road - model 2.

Figure 7.26 shows the oxygen distribution under this situation for model 3 representing operating longwall retreated close to the finish off line of the longwall block. Again it can be observed from the modelling results that air ingress into the goaf behind the sealed longwall blocks will be significantly increased, in particular for LW1-LW6.
Figure 7.26 Gas distribution pattern in goaf with roof fall in the perimeter road - model 3
7.5.3 Spontaneous combustion gaseous products flow patterns

R70 testing of Mandalong coal from the West Wallarah Seam has indicated that there is a potential for spontaneous heating. Ethane \((C_2H_6)\) has been detected as a seam gas at Mandalong mine and testing has indicated that ethane liberation at low temperatures can be varying. To assist the management of sponcom at Mandalong mine and in particular the early detection of a likely sponcom oxidation event, the behaviour of CO and \(C_2H_6\) has been investigated in this study.

Based upon the base model results and goaf gas monitoring data, it is assumed the most likely areas liable to sponcom development would be in the active goaf, the areas around the start-up seals of LW1-2 and LW4-5 and the adjacent goafs of LW7-8 at the middle part of the sealed areas. In all cases, a constant gas evolution value of \(10l/s\) is assumed for CO and \(C_2H_6\) from these locations and the heating effect is ignored for simplicity.

Figure 7.27 shows the modelling results of CO flow patterns in the goaf areas as it evolves from different locations of a potential heating site. Modelling indicates that CO gas from these locations is likely to disperse deep into the goaf and eventually appear onto the seals of the Perimeter Roadway along the TG side of LW1.

Figure 7.28 shows CO and \(C_2H_6\) flow pattern in the goaf from a potential heating site around the start-up areas of LW4-5. A similar pattern for both CO and \(C_2H_6\) can be observed with most of the gaseous product dissipating into the goaf and gradually to appear along the seals of the perimeter road along the TG side of LW1. The modelling results did not show any significant difference in the flow behaviour of CO and \(C_2H_6\).

Figure 7.29 shows CO and \(C_2H_6\) flow pattern in the goaf from a potential heating in active goaf on the MG side of LW9. A similar pattern for both CO and \(C_2H_6\) can be observed with most of the gaseous product dissipating into the deep goaf and eventually appearing along the seals of the perimeter road close to the Mains at highly diluted levels. The modelling results also indicate that it would be difficult to
detect an active goaf heating (on MG side) at its earlier stage by simply relying upon CO readings in the return airflow, as the main stream of the gaseous product will be flowing into the sealed deep goafs in adjacent LW panels. It is therefore important to station monitoring points (such as tube bundles and bag sampling points) at deep seated seals along the active goaf so that abnormal CO or $\text{C}_2\text{H}_6$ readings can be picked up for early detection and accurate location of potential heating spots.

Figure 7.27 CO flow patterns in the goaf area

(a) CO from LW3-4  (b) CO from LW1  (c) CO from LW7-8 middle part

Figure 7.28 CO and $\text{C}_2\text{H}_6$ flow patterns in the goaf area
Figure 7.29 CO and C\textsubscript{2}H\textsubscript{6} gas flow patterns in the goaf area - heating in active goaf

7.5.4 Impact of face ventilation on CO and C\textsubscript{2}H\textsubscript{6} flow pattern

CFD modelling was conducted to investigate the impact of face ventilation changes on the dispersion pattern of sponcom gaseous products. As CO and C\textsubscript{2}H\textsubscript{6} behave in a similar manner, only the results on CO are presented here for simplicity.

Figure 7.30 shows the CO dispersion pattern from an assumed heating source in the active goaf and adjacent sealed goaf when the face ventilation was increased by about 20 m\textsuperscript{3}/s (perimeter road ventilation was reduced by an equivalent volume). Results show that CO dispersion in the active goaf becomes much extensive with diluted concentration moving towards the deep goaf and the perimeter road, thus makes the heating difficult to detect and locate; whilst for the deep seated heating in the adjacent sealed goaf, CO dispersion is to be pushed further into the deep goaf, making it almost impossible to be detected by tube bundles or bag sampling around the goaf seals.
Figure 7.30 CO dispersion pattern with increased face ventilation

Figure 7.31 shows the CO dispersion pattern from an assumed heating source in the active goaf and adjacent sealed goaf when the face ventilation was reduced by about 20 m$^3$/s (perimeter road ventilation increased by an equivalent volume). Results show that CO dispersion in the active goaf becomes confined and localised with elevated concentration around the seals on the MG side and eventually reports to return ventilation and adjacent goafs, consequently it can be picked up by tube bundles and bag samples via goaf seals behind the longwall and gas detectors in tailgate. For the deep seated heating in the adjacent sealed goaf, CO dispersion is likely to be drawn back towards the tailgate and appear in the seals along the tailgate and the Mains.

Figure 7.32 shows the pattern of CO dispersion from a heating on the edge of the sealed goaf close to the perimeter road in response to increased or reduced face ventilation. Modelling results indicate that increased face ventilation will promote the gaseous products to appear around the seals in the vicinity of the heating at elevated concentration and can be picked up by bag sampling whilst reduced face
ventilation allows the gaseous products to dissipate into the sealed goaf areas and eventually appear via seals along the perimeter road close to the Mains.

(a) CO from active goaf heating areas  (b) CO from sealed deep goaf

Figure 7.31 CO dispersion pattern with reduced face ventilation
(a) Face ventilation quantity increased  
(b) Face ventilation quantity reduced

Figure 7.32 Impact of ventilation on CO dispersion from a heating on the edge of sealed goaf

7.5.5 With regulator applied in new LW TG Active Goaf upon Sealing

Base Model 3 was used to simulate the effect of a new TG regulator on active goaf sealing. Figure 7.33 illustrates the ventilation system when the operating longwall (LW9) has retreated to the finish-off line and the new tailgate for LW10 has formed and a regulator installed. CFD modelling was conducted to evaluate the impact of the regulator on goaf gas flow pattern.

Figure 7.34 shows the goaf gas distribution pattern following the use of regulator applied in the new LW TG. Modelling results indicate that the installation of a ventilation regulator in the LW TG will help draw goaf gas towards the face and seals on the MG side of the near-finish off panel thus helping the self-inertisation process of the completing longwall panel. However the modelling also indicate that as much of the goaf gas (methane) has been pulled towards the face, increased air
leakage is likely to occur at the back seals of the old goafs particularly at LW7-8 and LW3, where a zone of high level oxygen is forming potentially leading to goaf oxidation. Caution is therefore needed to closely monitor and tighten up seals around these areas.

Figure 7.33 Ventilation scheme with regulator applied in the new LW TG upon sealing
Figure 7.34 Goaf gas flow pattern with regulations applied in new LW TG upon sealing.
7.5.6 Sealing flank and effect upon sealed goaf

In order to improve goaf inertisation, it was considered to seal the Flank upon seal-off the goaf. Base Model 3 was used to study the effect of sealing flank on goaf gas behaviour. Figure 7.35 shows the layout of the ventilation system and the position of the proposed seal in the perimeter road. Figure 7.36 shows the modelling results which indicate that the sealing of the perimeter road can reduce oxygen penetration into the active goaf and help the build up of methane in the sealed goafs and therefore improve the self-inertisation process.

Figure 7.35 Ventilation system while sealing the flank
7.5.7 Flood ventilation of flank/perimeter road

Base Model 3 was also used to study the effect of flood ventilation of flank/perimeter road on goaf gas behaviour. Figure 7.37 shows the schematic diagram of ventilation
system for this study in which the ventilation rate for the perimeter road was increased to 54 m$^3$/s. Figure 7.38 shows the distribution of oxygen in the sealed and active goafs following the flooding ventilation. Modelling results indicate that the flooding of the perimeter road can help push the build up of high concentration methane into the deep goaf areas in particular in active goaf, however this method is likely to induce excessive air leakage in certain parts of the sealed goaf, particularly around the seals of LW1-4 in the start-up area.

Figure 7.37 Flood ventilation of flank/perimeter road
(a) Methane

(b) Oxygen

Figure 7.38 Goaf gas distribution following flooding ventilation of flank/perimeter road
7.5.8 Goaf inertisation using nitrogen (N$_2$)

Base Model 2 was used to simulate the effect of nitrogen injection at various locations behind the active goaf. Nitrogen injection at 80 m and 200 m behind the face was modelled respectively. Figure 7.39 shows the effect of goaf inertisation by injecting nitrogen at different injection points behind the longwall face (e.g., LW9). An injection rate of 0.5 m$^3$/s is assumed in these modelling studies. Modelling results show that it is far more effective to inert the goaf by injecting nitrogen via a deep cut-through seals, i.e., at least 200 m behind the face, rather than at a point too close to the face, from where much of the injected inert gas will be dissipated by air leakage thus reducing its inertisation effect. Modelling results also indicate the importance of tight seals inbye from injection point as air leakage (oxygen ingress) will compromise the effect of inert gas even at deep injection points.
7.5.9 Goaf inertisation using in-seam drainage methane

In-seam methane drainage has been used at Mandalong mine to reduce rib emissions during longwall mining. It has been proposed that this high purity methane can be pumped into the sealed goaf and help inertise the goaf areas. CFD modelling was used to simulate the effect of injecting methane gas in both active goaf and sealed goaf.

Figure 7.40 shows the injection of methane (assuming injection rate at about 800 l/s) into the active goaf at about 200 m behind the face. Modelling results indicate that much of the injected methane will migrate into the deep parts of the goaf areas and in particularly to higher elevation areas due to the buoyancy effect.

Additional modelling results also show that injecting a low flow rate of methane (i.e. below 200 l/s) into the goaf will have negligible effect on goaf inertisation at working levels, as much of the injected gas will tend to migrate to higher elevations due to buoyancy effect.
(a) Methane

(b) Oxygen

(c) Methane - 3D view
7.6 Summary

This chapter describes the application of the CFD models to investigate the behaviour of goaf gas flow in both active and sealed goaf areas. General information of the longwall goafs and field monitoring data were collected from Mandalong Mine, based on which the CFD models were built to represent the goaf situations when the active longwall (LW9) retreats at start-up, middle stage and near finish-off lines. Results from these base models were calibrated and compared against field goaf gas monitoring data with good agreement. The base models were then used to carry out parametric studies to investigate a number of operational scenarios and their impact on goaf gas behaviour. In summary, the following observations could be made from the modelling results:

- Overall goaf gas flow patterns change as the operating longwall (e.g. LW9) retreats from the start-up position to finish-off line. This is mainly due to the changing ventilation pressure differential across the entire goaf areas as well as the varying goaf elevations along the longwall panels;
- At the start-up stage, high methane level tends to build up at those higher elevation areas close to the Perimeter Roadway and deep goaf areas towards the Mains, however oxygen ingress into the goaf is high (up to 18%) in the active goaf and the around the sealed areas around the start-up lines of LW1-2 and LW4-5;

- As the longwall retreats towards the middle part of the panel, oxygen ingress will increase significantly into the sealed goafs of adjacent longwalls (LW7-8), i.e., the lowest parts of the goaf area, and the methane level tends to build up at increased concentrations beyond this area, in particularly in zones close to the Mains and the higher goaf edges long the Perimeter Roadway;

- When the longwall reaches the near finish-off line of the panel, methane distribution in the goaf gradually becomes more regular throughout the sealed goafs and oxygen ingress is largely confined to the active goaf and some areas around the start-up seals in LW1-4;

- In all cases, oxygen penetration into the active goaf remains high, reaching 15% or above even some 800 m behind the longwall face;

- Oxygen ingress into the goaf would be more serious if roof fall or roadway failure in the perimeter road occurs and restricts ventilation, indicated by the increasing trend of air penetrating through the cut throughs before the roof fall position;

- The most likely areas liable to sponcom development would be in the active goaf, the areas around the start-up seals of LW1-2 and LW4-5 and the adjacent goafs of LW7-8 at the middle part of the sealed areas;

- A similar pattern for both CO and C$_2$H$_6$ can be observed with most of the gaseous product dissipating into the goaf and gradually appearing along the seals of the Perimeter Roadway along the TG side of LW1;

- It would be difficult to detect an active goaf heating (on the MG side) at its earlier stage by simply relying upon CO readings in return airflow, as the main stream of the gaseous product will be flowing into the sealed deep goafs in adjacent LW panels. Monitoring points (such as tube bundles and bag sampling points) should be selected at deep seated seals along the active goaf so that abnormal CO or C$_2$H$_6$ readings can be picked up for early detection and location of potential heating spots;
• Gaseous products such as CO flow pattern is sensitive to the change of face ventilation, with increased face ventilation, CO dispersion in the active goaf becomes much extensive with diluted concentration towards deep goaf and the perimeter road; whilst for the deep seated heating in the adjacent sealed goaf, CO dispersion is to be further pushed into the deep goaf, making it almost impossible to be detected by tube bundles or bag sampling around the goaf seals.

• Regulation in the LW TG will help draw the goaf gas towards the face and seals on the MG side of the near-finish panel thus helping the self-inertisation process of the completing longwall panel; the sealing of the perimeter road can also help the build up of methane in the sealed goafs;

• Flooding of the perimeter road can help push the build up of high concentration methane into the deep goaf areas in particular in the active goaf, however this method is likely to induce excessive air leakage in certain parts of the sealed goaf, particularly around the seals of LW1-4 in the start-up area;

• Goaf inertisation can be better achieved by injecting inert gas such as nitrogen at deeper points (>200 m) behind the operating longwall; the injection of in-seam drainage methane into the goaf areas will only have limited effect on goaf inertisation as much of the injected methane will migrate towards deep and higher parts of the goaf due to its buoyancy effect.

Throughout the studies conducted in this chapter, it can be concluded that: 1) the CFD models can be used to investigate the gas flow patterns in large goafs areas in combination with field gas monitoring data; 2) compared with field operational practice, it is much safer and more cost effective to carry out parametric studies so as to investigate the impact of operational changes on goaf gas distribution and migration characteristics; 3) findings obtained from this study are of great importance to the management of large goaf areas in underground coal mines; 4) the CFD models should be constructed according to field conditions, i.e., the floor elevation of the large goaf area must be considered in this study, otherwise, some important information cannot be obtained with the general flat models.
8 IMPROVED RESPIRABLE DUST CONTROL FOR DUST FROM THE MG CHOCKS AND BSL

8.1 Introduction

Fugitive dust on longwalls has always been an issue of concern for production, safety and the health of workers in the underground coal mining industry globally. Respirable dust particles have long been known to be a serious health hazard to workers in coal mining. Prolonged exposure to excessive levels of airborne respirable coal dust can lead to Coal Workers’ Pneumoconiosis (CWP), Progressive Massive Fibrosis (PMF), and Chronic Obstructive Pulmonary Disease (COPD). These diseases are irreversible and can be debilitating, progressive, and potentially fatal in their most extreme cases.

Dust particles can be generated from several sources on the longwall face, primarily shearer cutting, chock movements, stage loader/crusher and intake contaminations. The drive for high production output has led to the advancements in longwall technologies with more powerful and faster shearers, and these consequently require chock movements at a faster rate. As the supports are lowered and advanced, crushed coal and/or roof rock drops from the top of the canopy into the airflow on the longwall face. As a result, chock movements can be a significant source of respirable dust for shearer operators when chocks are advanced upwind of the shearer during the MG to TG cut.

Gillies and Wu (2007; 2008; 2010) conducted extensive real-time respirable dust monitoring and baseline surveys on Australian longwalls. The results from a LW face baseline survey located in the Bowen Basin, Queensland, are shown in Figure 8.1. During the cutting sequence from MG to TG, 1-5 MG Chocks were advanced immediately after the shearer passed. This action leads to much dust falling from the advancing chocks; dust levels registered by Personal Dust Monitor (PDM) units were increased significantly, accounting for about 47.8% of total longwall dust make at
the shearer MG operator’s position during the cutting cycle, as indicated in Figure 8.1. Similar observations have also been reported at other mines in Australia.

Figure 8.1 Dust survey on a longwall showing MG Chocks (1-5) advance dust versus total longwall dust make at shearer MG and TG operator positions (Gillies and Wu, 2008, 2010)

The dust baseline survey results demonstrated the importance of reducing respirable dust generated from the advancement of MG chocks thereby significantly mitigating
total dust make to the longwall face. Dust monitoring also showed that the BSL can be another major dust contributor, even for longwalls equipped with a BSL dust scrubber. Dust surveys indicated that the scrubber can cleanse only a portion of the air travelling to the face, allowing much of the dust particles to escape over the BSL and to end up on the longwall face, increasing the threshold dust levels in the ventilation air.

Effective dust control is important for occupational health and safety of production crews and eventually production outputs. With the use the newly developed ultra-fine water mist venturi system, detailed CFD modelling studies of the airflow-dust particle dispersion patterns from MG chocks and the BSL were conducted to determine the optimum positioning of the water mist venturi unit(s) for effective dust control. Then, according to the modelling results, field trials were implemented at Metropolitan colliery. The performance of the dust suppression system which has been evaluated and assessed by real time dust monitoring during the trial is also presented in this chapter. It is noted that this chapter relied heavily on the ACARP C18019 report, where some more details about the design of the water mist unit(s) can be found, as well as the validation of the CFD models (Ren, et al., 2012).

8.2 The water mist based venturi unit(s) for respirable dust mitigation at longwall faces

8.2.1 Respirable dust capture mechanism using the water mist technology

Historically water sprays have been used for dust control in the mining industry to prevent the dust particles from becoming airborne (Goodman, 2000; Colinet, et al., 2010). However, field practices have demonstrated that these dust control systems are not effective for respirable dust mitigation (Goodman, 2000; Pollock and Organiscak, 2007). Research outcomes of Schowengerdt and Brown (1976) indicated that effective respirable dust suppression was achievable if sprays were able to produce droplets with comparable size to the respirable dust particles, which would allow a droplet to impinge on a dust particle, and thus heavy agglomerates of dust
and water could be formed progressively, resulting in a "settling out" of the airborne
dust. Conventional sprays generating water droplets with typical diameters of 200-
600 μm are not effective on respirable dust because the droplets are too large to
collide with the finest, most hazardous dust particles smaller than 10 μm. Attractions
between airborne dust particles and water droplets are more likely to occur when
they are of similar size.

From the perspective of fluid-particle two-phase flow, this phenomenon could be
further interpreted. Stokes number, which governs the behaviour of particles in the
fluid in which they are immersed, must be employed to investigate whether the dust
particles move synchronously with the fluid streamlines. It is defined as the ratio of
particle response time (τ_p) to a time characteristic of the fluid motion (τ_f) (Crowe,
et al., 2011). When the flow passes water droplets, τ_f can be written as:

$$\tau_f = d_w / v_r$$

(8.1)

Therefore, the Stokes number can be expressed in the following form:

$$S_t = \frac{\tau_p}{\tau_f} = \frac{\rho_p d_p^2 v_r}{18 \mu d_w}$$

(8.2)

where, d_w is the diameter of water droplet, v_r is the relative velocity between dust
particles and droplets, ρ_p is the density of particle, d_p is the diameter of particle,
and μ is the dynamic viscosity of air, which is 1.81×10^{-5} Pa·s at 15.0°C.

It can be seen from Eq. 8.2 that for a specific size of respirable dust particle, the
Stokes number is only related to the relative velocity between particles and droplets
and the diameter of water droplets. In the case studied, it is reasonable to assume that
the respirable dust particles own the same velocity as the flow field, thus it can be
inferred that the relative velocity will not affect the order of Stokes number which is
more sensitive to the diameter of water droplets which may vary from several micron
to several hundred micron.
Consequently, it can be estimated from Eq. 8.2 that, if water droplets and dust particles are of similar size, the Stokes number will be much larger than one, thus the particles will have essentially no time to respond to the fluid velocity changes and inertia is more important, which implies that the dust particles will probably collide with the droplets rather than following the fluid streamlines; therefore, the collision probability is greatly increased between similar sizes of dust particles and droplets. Whilst if water droplets are much larger than the dust particles they are attempting to suppress, the Stokes number will be much smaller than one, which suggests that the particles have sufficient time to follow the fluid streamlines closely and no contact will occur. A detailed illustration showing the impact of droplet size on the capture of respirable dust particles can be found in Figure 8.2.

![Diagram showing the impact of droplet size on the capture of respirable dust particles](image)

Figure 8.2 The impact of droplet size on the capture of respirable dust particles (after Schowengerdt and Brown, 1976)

8.2.2 Prototype of the newly developed water mist based venturi unit(s) for longwall chocks

Figure 8.3 shows the first prototype water mist venturi systems that are ready for field trials. This new venturi system essentially consists of a water mist generating chamber incorporating mounting holes via which water and compressed air can be introduced to the ultrasonic atomisers that produce very fine droplets. Water is ejected through a number of orifices into the nozzle air outlet channel, where the
high velocity air stream produces a first liquid breakup through shear action. The air stream, carrying the droplets, collides with a resonator placed in front of the nozzle outlet channel that generates a field of high frequency sound waves. Water delivered to the resonator is shattered into fine droplets which are then carried downstream by air by-passing the resonator.

The novelty of this new system is its capability to draw sufficient air into the chamber to carry the atomised droplets downstream with sufficient momentum for maximum dust particle attraction and controlled diversion away from the walkway area. In order to utilise the existing water and compressed air supply on the longwall face, the system is built as a stand-alone module with a magnetic base which can be easily attached to the chocks’ canopy and adjusted with the right spray angle to achieve the droplet size and velocity needed for dust suppression and diversion.

![Image](image_url)

Figure 8.3 The completed prototype water mist venturi units

### 8.3 Equations governing the respirable dust flow

The dispersion of respirable dust particles can be determined by either the Euler-Lagrange or the Euler-Euler method. The movement of particles is described by tracking a large number of particles through the calculated flow field in the Euler-
Lagrange method where the interactions between particles are neglected; while the discrete dust particles are treated as interpenetrating continua when the Euler-Euler method is employed (ANSYS, 2010). A comparison of the two methods indicated that the Euler-Lagrange method is more suitable for this study as the volume fraction of dust particles is much less than 10% (ANSYS, 2010).

With use of the Euler-Lagrange method, the trajectories of individual particles can be calculated by solving the momentum equation. By integrating the force balance on a particle, the momentum equation can be written in the following form:

\[
\frac{d\vec{u}_p}{dt} = F_D(\vec{u} - \vec{u}_p) + \frac{g(\rho_p - \rho)}{\rho_p} + \vec{F}
\]  

(8.3)

In Eq. 8.3, the left hand side stands for the inertial force per unit particle force, where \( \vec{u}_p \) is the particle velocity vector; the first term on the right hand side stands for the drag force per unit particle mass; the second term stands for the gravity and the buoyancy, where \( \rho \) and \( \rho_p \) are the density of fluid and particles. The last term stands for additional forces, which includes forces such as the virtual mass force, the thermophoretic force which is caused by temperature gradient, the pressure gradient force, the Brownian force and the Saffman’s lift force.

In this study, the dust particles are assumed to be spherical and the drag force is thus assumed to follow the spherical drag law, which can be expressed as:

\[
F_{\text{drag}} = F_D(\vec{u} - \vec{u}_p) = \frac{18\mu}{\rho_p d_p^2} \frac{C_D R_e}{24} (\vec{u} - \vec{u}_p)
\]  

(8.4)

where, \( C_D \) is the drag coefficient and can be calculated by the following equation \( C_D = a_1 + a_2 / R_e + a_3 / R_e^2 \), and \( a_1, a_2, a_3 \) are constants; \( R_e \) is the relative Reynolds number, which is defined as \( R_e = \rho d_p |\vec{u}_p - \vec{u}| / \mu \).
It is worth noting that the virtual mass force is important when the fluid density is larger than the particle density which is not the case of this study, and the virtual mass force is therefore neglected. Meanwhile, this study is conducted under isothermal condition, so the thermophoretic force and Brownian force are thus not considered either.

Therefore, by substituting the additional force $\vec{F} = \vec{F}_f + \vec{F}_s$ in Eq. (8.3), the particles can be tracked by solving the following equation:

$$\frac{d\vec{u}_p}{dt} = \frac{18 \mu}{\rho_p d_p^2} \frac{C_p R_e}{24} (\vec{u} - \vec{u}_p) + \frac{g(\rho_p - \rho)}{\rho_p} + \vec{F}_f + \vec{F}_s$$ (8.5)

where $\vec{F}_f$ and $\vec{F}_s$ stand for the fluid pressure gradient force and the Saffman’s lift force, respectively.

### 8.4 CFD modelling of respirable dust flow patterns

In this section, the longwall model is constructed on the basis of longwall geometry at Metropolitan colliery, which is different from the one used in Chapter 4. Figure 8.4 illustrates the layout of the longwall model, which is incorporated with 103 chocks, a simplified shearer and AFC along the face. Meanwhile, both BSL and belt conveyor are equipped in the MG. It is also noticed that two rows of water sprays were also mounted under the chock canopy and the section of coal transfer point from AFC to BSL to investigate the performance of various dust control options in terms of their operating conditions. Generally, the longwall face has physical dimensions of 156.5 m (face length) $\times$ 50 m (MG length) $\times$ 3.5 m (face height), and a 10 m cut through is developed at 45 m outbye the face. The longwall shearer is located 10 m away from the MG.

Upon validation of flow field in the base model, the model was used to investigate the respirable dust flow patterns at the MG corner. As the focus of this section is located at the MG corner, the following results only include the first third of longwall
face and not much attention is paid to the respirable flow at the further downstream of the face.

Figure 8.4 Layout of the longwall model

Aiming at understanding the dust dispersion characteristics, groups of respirable dust were released as coal-hv (material) with particle sizes between 1-10 µm from potential dust sources. The particle size distribution at each of the releasing point was assumed to follow the Rosin-Rammler distribution function. The dispersion of particles due to turbulence in the continuous phase flow phase (air) was tracked using the Discrete Random Walk (DRW) model, which includes the effect of instantaneous turbulent velocity fluctuations on the particle trajectories through the use of stochastic methods. The fluctuating velocities are assumed to follow a Gaussian
probability distribution in the DRW model; and the fluctuating components can be defined as:

\[ u' = \zeta \sqrt{u'^2} = \zeta \sqrt{2k' / 3} \]  

(8.6)

where \( \zeta \) is a normally distributed random number and \( k \) is the turbulent kinetic energy.

By replacing the velocity \( \bar{u} \) with \( \bar{u} = \bar{u} + u' \) in the momentum equation, each trajectory could then interact with the modelled flow field at each instance in time.

Figure 8.5 shows the tracking of these particles released from the MG belt and BSL, and the gaps between chocks in the case of shearer cutting from MG to TG. Modelling results show that dust generated from the MG accounts for a large proportion of dust exposure levels for workers around the longwall control and communication panel, and this impact can extend to the shearer operators inside the face, meanwhile dust generated in the vicinity of the BSL also contributes large amounts of dust to the walkway. It is observed that the generation of dust particles due to chocks movement behind the shearer is another major contributor to shearer operators’ high exposure levels while cutting from MG to TG. To avoid the hazardous dust produced during chocks advancement, the unidirectional cutting sequence is adopted in many underground coal mines (Aziz, et al., 2009).

(a) Dust particles from MG belt
(b) Dust particles from BSL transfer position

(c) Dust particles from MG chocks

(d) Dust particles from chocks immediately behind the shearer

Figure 8.5 Flow patterns of dust particles released from MG, BSL and chocks-plan view

Figure 8.6 provides a 3D view of the particle tracks within the longwall face. It can be seen that the dust particles disperse widely along the face and there is considerable dust at the mining operators’ breathing level.
(a) Dust from the MG

(b) Dust from the BSL

(c) Dust from MG chocks
Based on a good understanding of the respirable dust dispersion behaviour from the base models results, two rows of water-mist sprays were mounted under the chock canopy and the AFC/BSL spill plate respectively to investigate the dust suppression effects on these potential dust sources. The application of these sprays was simulated by setting one face of each spray as velocity inlet, and the sprays can be operated individually or in groups as each of them was defined separately. According to the location of different dust sources, the sprays were switched on/off accordingly. The performance of these sprays is discussed in the following section.

It is noted that in the following study, the water mist generated by the venturi units was modelled as ‘air spray’ as much of the spray would be of compressed air with a small portion of fogged water droplets. This also avoids the complexity of modelling multiphase flows which would require much computing power and time.
8.5.1 Dust mitigation from the MG

Regardless of the shearer position along the face, the intake air is always under the threat of MG dust contamination as long as the face is in production. As a result, sprays mounted under the first six chocks were switched on to cleanse the dust particle-laden intake air. Meanwhile, to obtain an optimum dust mitigation effect, various operating options were set up for these sprays in terms of adjusting the spray direction.

Figure 8.7 indicates the impact of these sprays on the dust dispersion pattern owing to the change of regional airflow patterns at the MG corner. It can be seen that the results are quite variable as sprays are oriented to different directions, and in some cases the situation is made worse by diverting more dust particles into the walkway. A comparison of the results demonstrates that a good dust suppression result can be achieved when this group of sprays are oriented at 30° down and 20° along face.
(c) Operated at level and tilted $45^\circ$ along face

(d) Operated at $20^\circ$ down and tilted $30^\circ$ along face

(e) Operated at $20^\circ$ down and tilted $45^\circ$ along face

(f) Operated at $30^\circ$ down and tilted $20^\circ$ along face

Figure 8.7 The MG dust mitigation effect
8.5.2 Dust mitigation from BSL

To capture the dust particles generated from the BSL, sprays mounted on the spill plate at the MG corner were utilised. As indicated by the base model, the dispersion patterns of dust particles are highly associated with the local airflow patterns. Due to the impact of flow separation, the dust particles are involved in the circulation above the coal transfer point and are blown up above the top of spill plate, dispersing into the face. Once the dust dispersion mechanism is known, it becomes much easier to solve it.

As shown in Figure 8.8, four sprays installed at the lower level were switched on to minimise the effect of flow separation and prevent these particles from going up and becoming airborne. Figure 8.9 illustrates the particles trajectories after the utilisation of these sprays. Compared with the result shown in Figure 8.5b, under these operating conditions, the number of particles succeed to flow into the walkway is significantly reduced, except the one when the sprays are operated horizontally. Therefore, the best performance can be achieved if the sprays are oriented between 30° and 40° down to the floor when most particles are blown to the face side without dispersing over the spill plate.

Figure 8.8 Sprays used for BSL dust mitigation
Figure 8.9 BSL dust mitigation effect
8.5.3 Dust mitigation from MG chocks

Base model results revealed that dust generated between the gaps of MG chocks will pass through the communication panel with airflow to the back of chock, and finally appears at the walkway next to the shearer body. Thus, the second row of sprays mounted to the canopy was also taken into action to assist sprays of the first row. Figure 8.10 is a snapshot showing the sprays involved in the MG dust mitigation.

Figure 8.11 illustrates the influence of these sprays on the dust particles dispersion trajectory. It can be seen that there are still quite a large number of particles appearing in the walkway even though some of them have been blown to the face rib side. Therefore, the key issue in dealing with respirable dust lies in how to capture those particles penetrating through the mist. Now that it is difficult to stop the particles penetrating to the back of chocks, a better mitigation effect could be obtained if the particles could be trapped at the back of chocks and stopped going any further to the walkway. Comparatively, sprays operated at 30º down and 20º along face can give better dust mitigation results than other operating conditions.

Figure 8.10 Sprays used for MG chock dust mitigation
(a) Operated at level

(b) Operated at level and titled 20° along face

(c) Operated at level and titled 45° along face

(d) Operated at 20° down and titled 30° along face
8.5.4 Dust mitigation from chocks immediately behind the shearer

Base model results revealed that dust generated by the chock movement immediately behind the shearer contributed significantly to the shearer operators’ dust exposure level, as a result, another six sprays were utilised to keep this dust away from the operator’s location. Figure 8.12 illustrates the position of sprays involved in this case and their impact on the respirable dust dispersion patterns. It can be seen that dust particles are forced to disperse towards the face side at different degrees within various distance to the TG side. A comparison of Figure 8.7f and Figure 8.12f indicates that the best operating condition for dust suppression from the MG is not suitable for dust from the chocks in the face, more sprays need to be switched on as some particles come back to the walkway again. Therefore, considering the distance that the particles can be kept away from walkway, sprays should be operated at 20° down and tilted 45° along face.
(a) Operated at level and perpendicular to face

(b) Operated at level and tilted 20° along face

(c) Operated at level and tilted 45° along face

(d) Operated at 20° down and tilted 30° along face
Field trials were conducted at Metropolitan Colliery of NSW, Australia and dust surveys were conducted concurrently to evaluate the dust mitigation efficiency of the venturi units installed on the MG Chock and BSL. A benchmark test was undertaken specifically designed to measure the amount of dust produced during chock movement in the MG area as the longwall progresses with further sampling taken for venturi sprays installed at the MG (BSL) and at Chock #6. It is noted that the longwall was vented with an air flow rate of 45m$^3$/s during the field trial.

8.6.1 Installation of the venturi units

Four venturi units were delivered to Metropolitan Colliery and installed for field trials, of which three units were installed on Chock #6, and one unit on the MG BSL.
Figure 8.13 and Figure 8.14 respectively show the installation of the venturi units on the BSL in the MG and longwall Chock #6. Figure 8.15 shows the units in operation during the trials. As shown in the diagram, these units were installed just under the canopy forehead and tilted along the face at angles between 15-20° down and 45° along the face, as indicated by the CFD modelling.

Figure 8.13 A venturi unit installed on the BSL in the MG

(a) Venturi unit installed on longwall Chock#6 (wires are for earthing)
(b) Venturi unit installed on longwall Chock#6 (wires are for earthing)

(c) Venturi unit installed on longwall Chock#6 (showing the magnetic disk attached to the canopy)
(d) Venturi unit installed on longwall Chock#6 (showing the manifold controlling water and air supply to the venturi)

Figure 8.14 Venturi units installed on MG Chock#6

(a) Venturi units in operation
Air and water supplied to these units was around 6 bar and 3.5 bar respectively, with a water consumption of 2 l/min per unit (total of 6 l/min for three units). The induced airflow velocity at the outlet mouth of the venturi unit would be around 8 m/s according the lab test results, thus having some momentum for diverting and streamlining respirable dust clouds off the walkway area along the face.

8.6.2 Dust measurement results

The gravimetric method was used to assess the efficiency of the installed venturi units, and the time interval selected during the test was a cutting cycle rather than the entire shift, which indicates that the respirable dust is collected over the period of one shear for each of the tests.

Figure 8.16 shows the location of the dust monitor placement along the longwall face, and the marked numbers represent the position of chock #2, #5, #8 and #15 respectively. It can be seen that one dust monitor was installed on chock #2, which could be used to investigate the efficiency of venturi unit on the BSL. As three
venturi units were installed on chocks #6, two dust monitors were placed on chock #5 and #8 respectively to determine the dust level directly before and after these sprays. Meanwhile, one monitor was placed on chock #15 to determine the sprays’ impact further downstream of the longwall. As a result, the control efficiency of the venturi units will be indicated by the weighed difference.

![Figure 8.16 Location of dust monitor placement along the longwall face](image)

For the purpose of comparison, the first test was taken as a benchmark test which was conducted during one shear with the existing controls operating and all the venturi units off. It is noted that one shear is able to produce 750 tonnes of coal at Metropolitan Colliery. The second test was carried out with the BSL venturi spray operating during another shear. The third test was undertaken when the chock sprays were turned on within another cutting cycle. It is worth noting that the gravimetric heads were changed during each testing process.

Figure 8.17 shows the respirable benchmark dust production at the proposed chocks along the longwall face during normal longwall production and without the operation of venturi units.
Figure 8.17 Respirable benchmark dust production

Figure 8.18 illustrates the dust production at different longwall chocks when the BSL venturi was off and on. And no significant reduction in the respirable dust production can be observed at the four positions. Figure 8.19 summarises the corresponding respirable dust mitigation efficiency (denoted as DME in the diagram) by the BSL venturi unit, where it can be seen that the single venturi unit has reduced the dust production along the longwall face between 5% and 13%.

Figure 8.20 shows the dust production at different longwall chocks when the venturi units on the chock #6 were off and on. It is observed that at chock #5 and chock #8, the respirable dust production is greatly reduced, indicating the major impact of the chock sprays occurs in the vicinity of the sprays both upstream and downstream.

Figure 8.18 Respirable dust production benchmark versus BSL venturi on
Figure 8.19 BSL venturi spray respirable dust mitigation efficiency

Figure 8.20 Respirable dust production benchmark versus chock venturi’s operating

Figure 8.21 summarises the dust mitigation efficiency caused by the chock venturi units. As can be seen in Figure 8.21, at chock #8, the maximum respirable dust mitigation can be up to 27% in the immediate downstream of the sprays. However, the effect is limited further downstream at chock #15, where only 7% of dust reduction is calculated. Meanwhile, it is worth noting that the respirable dust level at chock #2 is also reduced by 7%, and the actual reason for this reduction cannot be determined through this one set of data. But it is inferred that the reduction may result from either the induction effect of chock sprays or activities conducted in the MG intake. Some more tests may be needed as a follow up study to further evaluate the dust mitigation effect and identify the reason leading to the reduction at chock #2.
Figure 8.21 The respirable dust mitigation efficiency by chock venturi units

The combined respirable dust mitigation efficiency of BSL spray and the chock sprays is depicted in Figure 8.22, where an overall dust mitigation efficiency up to 35% has been achieved at chock #5 and 16% down at chock #15.

Figure 8.22 Total respirable dust mitigation efficiency by the installation of venturi units at BSL and chock #6

The testing results imply that the venturi sprays have a significant effect on respirable dust in the MG area, more specifically from chock #2 to chock #15. With
the operation of the BSL venturi, a reduction of respirable dust by 12% at chock #2, 13% at chock #5, 5% at chock #8 and 9% at chock #15 was achieved.

Meanwhile, compared with the BSL venturi spray, the chock sprays operating at chock #6 demonstrated a greater effect on respirable dust in the same area. The operation of chock venturis resulted in a reduction of respirable dust by 7% at chock #2, 22% at chock #5, 27% at chock #8 and 7% at chock #15.

Consequently, the combined effect of both the BSL spray and the chock sprays operating indicated a total reduction of respirable dust by 19% at chock #2, 35% at chock #5, 32% at chock #8 and 16% at chock #15.

8.6.3 Field observations

The management team at Metropolitan Colliery has been strongly supporting the field trials and was happy to see the encouraging results. The longwall crew also welcomed the venturi system as a new tool for combating dust contaminations on their longwalls.

Field trials demonstrated that the design of the water mist based venturi units has been a success to reduce dust contamination from longwall chock movements and the MG/BSL. The positions of these venturi units, either on the BSL or MG chocks, are important in diverting and suppressing the dust clouds with minimum wetting side-effect on the longwall. The installation and adjustment of these units has been simple and straightforward without any disturbance to existing equipment or operation. Again these venturi units demonstrated their robustness and remained clearance of the floor and other equipment without being damaged during the trial.

However the issue of wetting by the water mist travelling along the face has also be raised. As shown in Figure 8.23, the water mist will travel some distance along the face, and at some point, the water mist cloud will be more widely dispersed across the face, although offering additional benefits of dust suppression and cooling, it
does however have the side-effect of wetting and in some case, causing poor visibility.

This problem can be minimised by positioning the units further forward under the canopy or turning off the units once the advance of the 1-5 MG chocks was completed. Again during the trials, the venturi units were turned on during the entire production cycle (even after the advance of MG chocks #1 to #5). Bear in mind that these units are designed to be used only during the advance period of these MG chocks and they should be turned off once the chock movement is completed. This would avoid the problem of over wetting and potentially the visibility problem.

Figure 8.23 The water mist cloud travelling along the face

8.7 Summary

Fugitive dust on longwalls has always been an issue of concern for production, safety and the health of workers in the underground coal mining industry. Dust particles can be generated from several sources on the longwall, including shear cutting, chock movements, stage loader/crusher and intake contaminations. As the supports are lowered and advanced, crushed coal and/or roof rock drops from the top of the canopy into the airflow on the longwall face. As a result, chock movements can be a significant source of respirable dust for shearer operators when chocks are advanced upwind of the shearer during the MG to TG cut. Chock movement close to the MG
has been identified as a significant source of dust exposure for longwall operators when chocks are advanced upwind of the shearer during the MG to TG cutting sequence.

CFD modelling studies have been conducted and showed that much of the dust particles will follow the ventilation into the longwall, contributing significantly to longwall workers’ dust exposure, if not controlled by effective dust mitigation methods. Modelling results demonstrated that much of the respirable dust particles generated from MG chock movements and BSL will disperse into the longwall face, contributing significantly to dust exposure levels by longwall operators.

A theoretical analysis showed that the respirable dust can be effectively captured by the ultra-fine water mist which is characterised by a comparable size to the respirable dust particles. Therefore, a new water mist generating venturi system has been designed and field trialled to mitigate dust particles from the MG chock movements. The novelty of this new system is its capability to draw air into the chamber to carry the atomised droplets downstream with sufficient momentum for dust particle attraction and controlled diversion away from the walkway area. The system is built as a stand-alone module using Fire Resistance and Anti-Static (FRAS) materials with a magnetic base which can be easily attached to the chocks’ canopy and adjusted with the right spray angle to achieve the droplet size and velocity needed for dust suppression and diversion. Field trials show that dust reduction efficiency over 30% has been achieved at the longwall face. However, the issue of wetting by the water mist travelling along the face has been raised. Therefore, it is suggested that the wetting problem can be minimised by positioning the units further front under the canopy or by turning off the units once the advance of the MG chocks is completed.

The potential for the application of this technology in other areas of mining can be enormous, particularly in underground coal mines and hard rock mines. Such units can also be deployed for dust mitigation in tunnelling, surface stockpiling and mineral processing plants. Improved design and further trials are also needed to improve its operation and dust mitigation performance.
9 RESPIRABLE DUST CONTROL ABOVE AN UNDERGROUND BIN

9.1 Introduction

The extraction, crushing, and transport of coal in mining operations can generate significant amounts of airborne respirable coal dust. In chapter 8, the respirable dust flow patterns and their control at a typical longwall face have been discussed. In this chapter, the generation of respirable dust and its flow patterns above an underground bin will be investigated, together with the design of an innovative dust mitigation system and its performance evaluation.

This study is based on a real scenario occurring in an underground coal mine which is located in the Mandalong Valley of the Newcastle Coalfield, Australia. The mine produces 5.5 million tonnes of coal each year from its underground longwall operations. After many years of underground mining, a complex ventilation system has been developed as shown in Figure 9.1, in which the position of the bin in the ventilation system is also indicated. To prevent the accumulation of potentially explosive methane gas in the bin, the ventilation system has been designed to allow a large volume of fresh air (23 m$^3$/s) to pass through the bin to the belt road. Figure 9.2 provides a closer view of the ventilation and roadway layout around the bin, where it can be seen that the MT01 belt roadway intercepts the underground bin and serves as the main roadway carrying the intake air to inbye underground workings. This is the actual configuration used in the mine and compromisingly the upcoming ventilation airflow becomes the main source of dust pickup and subsequent dispersion along the belt road.

The underground bin is an important coal transfer station in the entire coal transport system of the mine. All the coal extracted from the longwall face and development headings is transported to the MT01 belt roadway and then crushed and dumped to the bin before being lifted to the surface. The bottom of the underground bin is about 20 m below the floor of the belt roadway. Once the crushed coal blocks fall into the bin, a significant amount of dust particles are blown up by the upcoming airflow.
from the bottom of the bin, i.e, the large volume of rising airflow is the major factor leading to dust pickup and transport in the ventilation system. Figure 9.3 illustrates the dimension of the belt roadway and the infrastructure layout around the bin. It has been observed that the dust was mostly generated during production shifts when the coal was dumped on the sizer and subsequently dropped into the underground bin. These dust clouds then disperse into the belt roadway as air flows in from the bottom of bin.

Figure 9.1 Mine ventilation system (arrows showing the airflow direction)

Figure 9.2 Mine ventilation system around the underground bin
It should be noted that the operation and configuration of the bin in this underground coal mine is different from any other kind of bin/bunker in underground metal mines, or those in the surface stockpiles or storage facilities, in which passing-through ventilation does not normally occur. The underground bin in this underground mine, however, is about 20 m deep and 10 m in diameter, and requires the passage of a large quantity of ventilation (23 m$^3$/s) for the dilution of hazardous gas (i.e. methane) resulting from crushed coal, to ensure a safe operating environment. Regulation of this airflow would be useful for minimising the dust pickup however practically this is impossible due to the continuous operation of the bin, as the coal is fed via the conveyor belt which would run continuously during normal longwall coal production. Unlike the shallow or regulated bin/bunker used in underground metal mines or surface storage facilities, the influence of the airflow regimes generated from the falling coals within the bin is negligible in comparison with the large volume of air (23 m$^3$/s) entering from the bin bottom. The large intake airflow has been the major contributor to dust pick up within the bin and subsequent transport in the belt road. Therefore the study presented in this paper has been focused on the airflow and dust dispersion characteristics above the bin and along the belt road, rather than within the bin structure. There is a need to understand the complex ventilation airflow and dust dispersion patterns around the bin areas before a more effective dust control system can be designed and implemented on site.
9.2 Development of a CFD model and boundary conditions

Based on the information collected from the coal mine, a full-scale CFD model was generated to resemble layout above the underground bin. Figure 9.4 illustrates a snapshot of the geometry, boundary conditions and computational grid of the CFD model. The belt roadway is 6.0 m wide and 3.6 m high. The model was built as close as possible to the mine conditions by incorporating the key features around the underground bin, including the sizer and the belt conveyors. Simplifications were also made following the principle of not affecting the main flow field, for example, the fence and steel mesh on top of the bin which served as a walkway was not included in the computational model.

It is important to note that the bottom of the underground bin is 20.48 m below the floor of the belt roadway. Once the crushed coal blocks fall into the bin, regardless of what happens inside the bin, a significant amount of dust particles will be blown up by the upcoming airflow from the bottom of the bin and become airborne in the belt roadway, therefore, only 8 m of the bin immediately below the sizer was incorporated in the model. It is also noted that the diameter of the bin is 10 m, as a result, the impact of the displacement of airflow from falling coal (unlike bulk truck dumping, coal feeding to the bin is by belt in a more regular and continuous motion) within the bin on dust dispersion is negligible in comparison with the influence of the large volume of upcoming airflow. As the focus of this study is on the airflow and dust dispersion pattern above the bin and along the belt road, it is therefore not necessary to define a separate granular phase (a multiphase model) to study the minimum impact produced by falling coal particles on the primary ventilation airflow domain. Consequently, the focus of the CFD model has been to understand both the ventilation and respirable dust flow characteristics in the belt roadway and around the top of the bin where the workforce are working, and what is the quality of air provided to the mine workings through the belt roadway, all of which will be vital to the development of effective dust mitigation strategy.
(a) Plan view of the CFD model geometry

(b) Cross section view of the CFD model geometry

(c) 3D view of the CFD model geometry
Upon defining the geometry of the model, it was meshed using the tetrahedron method with over 1.43 million elements to accommodate its complex geometry. Model boundaries were categorised into different types during the meshing process so that different boundary conditions could be conveniently applied in the CFD solver. It can be seen from Figure 9.4c that fresh air is supplied to the domain through two inlets, one is from the small roadway on the right hand side (outbye) and the other is from the bottom of the bin. Finer grids were employed on top of the bin where the mixing of the two air streams occurred. Due to the low Mach number of the flow field, the flow medium (air) was assumed to be incompressible. Therefore, air velocity across the two velocity inlet boundaries can be calculated using the air quantity through these two intakes and the corresponding cross sections. The left hand side of the belt roadway constitutes the only outlet of the computational model and is defined as pressure outlet boundary condition. Other boundaries of the model, such as the roadway roof, floor, ribs and equipment surfaces were treated as standard walls. Considering the relatively high moisture content under the site conditions, the DPM boundary conditions on the standard walls were set as trap, indicating the termination of the particle motion. It is worth noting that the temperature difference of the real scenario in the domain described above is negligible and the CFD computation is thus carried out under isothermal condition.
9.3 Base model validation and results

9.3.1 Base model validation

As in previous chapters, the base model results must be validated against data from field measurements before accepting the model for parametric studies. In this study, due to the complexity of field configurations and the movement of various equipment above the bin, the area hosting the crusher and dumping facilities has been fenced off (No Go Zone) thus preventing any access for the measurement of velocity directly above the bin. It was therefore decided to use the relatively accurate velocity measurement in the belt roadway as part of the model validation. During a field ventilation survey the average airflow velocity values on six cross sections at 10, 15, 20, 25, 30 and 35 m inbye the bin along the belt roadway were measured using a handheld anemometer.

Given the boundary conditions, the basic airflow field could be obtained by solving the governing partial differential equations in a conservative form. Figure 9.5 shows a comparison of measured and predicted velocity profiles along the belt roadway. It can be seen that the CFD model predicted ventilation air velocity remains almost constant at 1.35 m/s with a negligible descending trend; whilst the field measured ventilation velocity data shows some marginal fluctuations due to the complex underground conditions and changes of roadway cross-sections. Both modelled and measured velocity profiles demonstrate a similar descending trend as the ventilation flows towards the outlet (underground workings).

During the ventilation survey, a zone of airflow circulation was observed in the walkway just inbye the coal crushing and drop point, as shown in Figure 9.6, where the monitored air velocity was 0.5 m/s in that area with almost no movement above it. Figure 9.7 provides the CFD predicted velocity vectors showing the airflow pattern as the ventilation from the bin entered the roadway. It can be seen that an airflow circulation area was predicted at exactly the same location as identified by field observations. Therefore, it can be concluded that this model has the ability to predict the underground airflow field around the bin and investigate the flow behaviour of
respirable dust in the underground belt roadway. The model predicted dust concentration values also showed a good agreement with field measured dust concentrations along the belt road, as described in section 9.3.4.

Figure 9.5 Comparison of predicted and measured airflow velocity profiles along the belt roadway

Figure 9.6 Airflow circulation area observed during field ventilation survey
9.3.2 Airflow patterns

The complexity of underground configurations for mounting the crusher over the bin makes field measurements difficult or inaccessible in some areas to capture the key features of airflow pattern. Although real time handheld electrical monitors have been used for underground ventilation survey, they still have difficulty in capturing details of the turbulent flow especially when moving mining equipment is involved. The CFD modelling approach provides advanced data visualisation of key airflow variables such as velocity contours and vectors at different cross-sections or levels, which it would otherwise not be able to be obtained by field measurements.

Figure 9.8 illustrates the velocity distribution pattern above the bin and along the belt road at different cross-sections in terms of velocity contour and vectors. It can be seen from the modelling results that as the air enters the roadway from the bin, it impacts on the various infrastructures mounted on the crusher and intercepts the fresh air from another intake (inlet 1), thus causing an extensive area of turbulence above the bin; most of the intake air from the bottom of the bin travels towards the belt roadway, while a small portion of the air is dragged towards the opposite direction - the retreating position for the sizer, where it meets the fresh air from the other intake on the right side, generating a zone of flow circulation, before turning
around to join the main stream of ventilation in the belt roadway. As the airflow leaves the area above the bin, it starts to accelerate gradually to a velocity at about 1.35 m/s as the roadway cross section area narrows down. The ventilation velocity contours clearly show the low velocity zones on the other side of the walkway (no access zone) above the bin, where slow airflow circulation and dust particle settlements are likely to occur. Figure 9.8c also indicates that airflow from the bin bottom is able to push the air stream coming from the right inlet towards the roof area. This effect is more clearly illustrated in Figure 9.9 where the air streamlines from the right inlet are shown. Therefore, it can be inferred that the above mentioned flow characteristic greatly weakens the dilution of fresh air to the dust-laden flow from the bin, contributing to the high dust concentration at the workers’ breathing level. Figure 9.10 provides a snapshot of air streamlines starting from A-A cross section where the generation of whirling motion of airflow can be clearly observed. The air streamlines also indicate that respirable dust particles following the air flow from the underground bin will be dispersed in the belt roadway, causing contamination to the fresh air to be supplied to inbye underground workings.

To further understand the turbulent characteristics of airflow field, the air velocity variations at four parallel elevations passing the four fixed points (A, B, C and D) along the belt roadway are illustrated in Figure 9.11. Modelling results show that the airflow velocity at elevations A and C (close to the roadway rib) are higher than that at B and D (close to the fence for the crusher) until the air flow reaches the middle area above the bin, after which the airflow velocity at the four levels increases dramatically from about 0.6 m/s to 1.8 m/s. When the airflow reaches the belt roadway, the impact of turbulence becomes significant, the airflow velocity at levels A and B becomes higher than that at C and D (close to the floor), and eventually the airflow velocity distribution becomes almost uniform as the ventilation travels towards the outlet (inbye underground workings).
(a) Airflow velocity distribution at 1.7 m above floor

(b) Airflow velocity distribution across the sizer
(c) Velocity distribution in the middle of walkway (1.5 m away from rib)

Figure 9.8 Airflow velocity distributions around the underground bin

Figure 9.9 Air streamlines indicating the flow of air stream from the horizontal intake
Figure 9.10 Streamlines showing the generation of airflow circulation

Figure 9.11 Airflow velocity profiles at fixed elevations across the computational domain
It can be concluded that the average velocity profile on continuous cross sections will not be able to reflect the impact of turbulence as it has been homogenised, while velocity profiles at fixed elevations (points) can depict the turbulence nature of airflow. This also implies that it is better to use average velocity profiles to calibrate the model while the velocity profile along a certain elevation across the computational domain can be used to probe into the turbulence of flow field.

9.3.3 Respirable dust dispersion patterns

The main source of dust in this case will be from the dropping of coal from the sizer to the bin. In this model, a cluster of respirable dust particles (particle sizes between 1~10 μm) were ‘released’ from a bounded surface which is 2.5 m under the sizer. The dispersion of particles in the airflow was tracked by using the stochastic tracking (random walk) model, which includes the effect of instantaneous turbulent velocity fluctuations on the particle trajectories through the use of stochastic methods. The fluctuating velocities are assumed to follow a Gaussian probability distribution in the DRW model.

Figure 9.12 shows the tracking of respirable dust particles generated from the surface 2.5 m under the sizer. As the flow of dust particles is highly dependent on the airflow patterns, most dust particles will follow the main airflow and travel towards the belt roadway, while a small number of dust particles disperse to the opposite direction from where they rise up from the bin and after a circulation travel, finally being dragged into the main airflow. It can be observed in Figure 9.12 that the dust particles will disperse widely in the belt roadway once they become airborne and appear at various elevations above the floor, contributing to high dust contamination of intake air in the belt roadway. The dispersion of dust particles from the underground bin is mostly dictated by the upcoming ventilation airflow pattern. To mitigate dust contamination to the intake air, the dust clouds from the bin need to be intercepted and suppressed well before they become airborne and dispersed in the belt roadway.
Based on the stochastic tracking model, the trajectory of each particle is calculated and visualised in Figure 9.12, however, these tracks only describe the dispersion of dust particles qualitatively. To illustrate its dispersion quantitatively, the concept of dust concentration is employed. This variable relates the model results to the widely used industrial standard. With the aid of this variable, the performance of the Lagrangian method can be better evaluated for understanding the behaviour of respirable dust particles.

As the stochastic particle tracking model with the Lagrangian method would introduce uncertainty and randomness, sufficient numbers of particles were employed to obtain a stable concentration field, making the stochastic tracking scheme statistically meaningful. Figure 9.13 provides a plot of the model predicted dust concentration versus field measured dust data along the belt roadway. It can be seen that with the increase of particle numbers being tracked, the predicted dust
concentration along the belt roadway becomes more stable with less variations. The results also indicate that when the number of tracked particle trajectories reached over 80,000, the predicted dust concentration matched well with field measured data. Figure 9.14 shows the predicted dust concentration contour at 1.7 m above the floor (the breathing zone). It can be observed that once the dust become airborne from the bin, it starts to disperse extensively in the area over the bin with a concentration of over 15 mg/m$^3$ in the walkway and further inside the coal dumping station. The dust particles then travel with the airflow causing extensive contamination to the entire ventilation in the belt road. Dust concentration varies between 3 mg/m$^3$ in the middle of walkway where dust particles are better diluted, and 11 mg/m$^3$ in areas close to the roadway rib and belt transport systems where the velocity is relatively lower in the belt road. Also referring to the velocity contour in Figure 9.8a, it can be seen that some dust particles are more likely to accumulate in areas just outbye the bin where the velocity is low or sluggish with circulation in the ventilation. The impact of airflow circulation on dust concentration can also be observed on the right hand side of Figure 9.14.

![Graph showing comparison of measured and predicted dust concentrations along the belt roadway.](image-url)

Figure 9.13 A comparison of measured and predicted dust concentrations along the belt roadway.
Figure 9.14 Predicted dust concentration contour at 1.7 m above the floor

Figure 9.15 depicts the evolution of respirable dust concentration at five cross sections along the belt roadway, which are located at 5 m, 15 m, 25 m, 35 m and 45 m outbye the bin, respectively. On each cross section, dust levels along four straight lines from the floor to roof with an interval of 0.6 m (as Figure 9.16 shows), are selected to evaluate the dust concentration distribution on that cross section. It can be seen from Figure 9.15 that the general dust distribution pattern is similar at the five cross sections, with relatively low concentration in regions close to roof due to the dilution of air stream from the right inlet; then as the elevation drops, the dust concentration increases, reaching the peak value at the breathing level around 1.5~2 m; after that the concentration decreases slowly until 0.4-0.5 m above the floor. It can also be observed that the maximum dust concentration decreases from around 9.5 mg/m$^3$ to 7.5 mg/m$^3$ as air travels away from the bin and the dust level variation between these lines narrows as well. This can be attributed to the turbulence of flow in areas close to the bin, as the air flows towards the mine inbye the dust distribution becomes homogeneous along the belt road as the air flow does. Additionally, there is still a certain amount of dust particles depositing on the floor so that it is necessary to maintain relatively high moisture content on the floor, and thus avoiding the re-entrainment of respirable dust particles in the intake air.
Figure 9.15 Evolution of dust concentration along the belt roadway (4.3 m$^3$/s from the horizontal intake)

Figure 9.16 Distribution of projected lines for dust concentration analysis

### 9.4 Impact of ventilation on respirable dust distribution

It can be seen from the base model that the dust-laden air from the bin constitutes the main contaminator of the ventilation system. Base model results also revealed that a good ventilation dilution for dust particles is achievable but constrained to the roof region using intake air from the right hand side. Therefore, a parametric study was
carried out to probe into the impact of ventilation on respirable dust concentration distribution in the domain. To avoid affecting the dilution of gas in the bin, flow rate from the bottom of the bin was maintained at the same value as the base model. Thus, the ventilation impact study was performed by means of increasing the flow rate from the horizontal intake while all the other boundary conditions remained the same as the base model, aiming at pushing the dust-laden air to a lower height level in the roadway and enhancing the dust dilution effect of the ventilation system.

Figure 9.17 illustrates the evolution of dust concentration along the belt roadway when the flow rate at the horizontal intake is increased to 7, 10 and 13 m$^3$/s, respectively. Bearing in mind base model results, the increase in flow rate will generate similar dust evolution history in the air flow direction which can be seen in Figure 9.17. What makes the difference with the base model is that the high concentration area progressively shrinks to a lower region within 1 m above floor, and the magnitude of dust concentration also decreases gradually from 9.5 mg/m$^3$ to 3 mg/m$^3$ at height levels between 1.5 m and 2.0 m (this range is assumed to be the breathing level of workers) when 13 m$^3$/s of air is ventilated to the domain. In other words, with gradual increase in flow rate, the air stream from the right intake is able to push the majority of respirable dust particles downwards to the floor level, and the dust concentration decreases as well. It is also observed that on each cross section, the high dust concentration area gradually migrates from the left side to the belt conveyor side as the flow rate increases, especially at the workers’ breathing level and above. Therefore, it can be concluded that significant improvement of ventilation dilution on respirable dust particles is achievable above the bin and further down the belt road if the flow rate from the horizontal air intake is increased to 10 or 13 m$^3$/s.
(a) Case 1 - 7 m³/s from the horizontal intake

(b) Case 2 - 10 m³/s from the horizontal intake
Design of an innovative respirable dust mitigation system

A good respirable dust mitigation effect can be achieved if the ultra fine water mist technology is correctly employed in a dust mitigation system. Field observations and CFD results confirmed that all respirable dust particles resulting from the bin will disperse into the belt road if no dust suppression system is installed on top of the bin to filter the dust-laden air. Based on the above understanding, a new water mist based dust mitigation system was proposed. As illustrated in Figure 9.18a, the new system consists of four sprays on each side of the sizer which are capable of producing the desired water mist. To better understand the potential operating effects of sprays illustrated in Figure 9.18b, the CFD models were again used to investigate the water mist flow behaviour for the design and implementation of the most effective dust mitigation system.

Figure 9.19 illustrates the model predicted water mist flow behaviour when the sprays are operating at different directions as proposed in Figure 9.18b. It can be
observed from the plan view that the general flow patterns of water mist are similar in terms of generating a good coverage on top of the bin, but with tiny difference when the mists get to the roadway. When all the sprays are directed with a certain angle downwards, the mists are more likely to spread across the entire section of the roadway, and this may cause visibility issues to workers in the walkway. A good coverage above the belt conveyor is also obtained which will impact the dispersion of dust particles from the transport system. Figure 9.19b shows the flow patterns of the mists under the sizer. When Group II sprays vertically down, the mists are well-distributed in the bin rather than involved in the flow circulation, and the velocity of generated mists is relatively low which will allow more time for the mists to collide with respirable dust particles thus enhancing the dust mitigation effect. The direction of Group I does not have a significant effect on the performance of Group II. Its main function is directing a fine curtain of water droplets inwards from the rim towards the crusher in the centre, and has the effect of containing and suppressing the dust being generated.

(a) Proposed locations of sprays for water mist generation

(b) Potential operating configurations of sprays

Figure 9.18 Proposed sprays location and potential operating conditions
Therefore, to achieve a good mixing effect between water mists and respirable dust particles, it is recommended that the sprays on the walkway side can be operated with a certain angle (smaller than 45°) downwards according to site configuration, and the sprays on the other side should be operated vertically down.
9.6 Field implementation and demonstration

9.6.1 Field implementation

Eventually, a new water mist based dust mitigation system was built and mounted on top of the bin as shown in Figure 9.20. The coverage of the nozzles as they spray across the sizer is also depicted in Figure 9.20. It is worth noting that the nozzles employed in the system were flat fan nozzles with standard capacity. Figure 9.21 shows the configurations of the nozzles selected for the new system.

According to the model results, the operating conditions of the nozzles in terms of spray direction are slightly different in order to achieve the best dust mitigation effect. The nozzles on the walkway side (Group I) were thus mounted to spray across the sizer with a small angle downwards and had a spray angle of 90 degrees which would give the spray more “strength” and therefore the generated mists were less prone to be blown away by the updraft. Working at 400 KPa, the flow rate of each nozzle was 9 l/min. While the nozzles installed on the other side of the sizer (Group II) were configured to spray vertically down with a spray angle of 120 degrees, and had a flow rate of 6.7 l/min per nozzle at 400 KPa. The selected system works by directing a fine curtain of water droplets inwards from the rim towards the crusher in the centre, and has the effect of containing and suppressing the dust being generated. Aided by gravity, the dust particles fall into the bin instead of remaining airborne. One significant advantage of the new system over the traditional air-atomising sprays is that the nozzle orifice is large enough that it will not be blocked easily, and therefore require no or minimum maintenance.
269

Figure 9.20 Design of the new dust mitigation system and its projected spray coverage

Figure 9.21 Configurations of the flat fan nozzle

9.6.2 Field demonstration

Following on-site installation of the dust suppression system, an independent dust measurement was conducted to assess the performance of the system. When the longwall face and headings were in normal production, the ‘TSI Dust Trak 2’ Aerosol Monitor was used for measuring the dust exposure levels along the belt roadway before and after the operation of the new system, respectively. The monitored dust result is given in Figure 9.22 and Figure 9.23. It can be seen that significant respirable dust reduction has been achieved along the belt roadway, especially at positions close to the bin, reaching a maximum reduction efficiency of 68%. On average, 40% of respirable dust particles have been successfully captured.
by the new water mist system. Field observations by mine operators have confirmed that this innovative dust mitigation system has performed satisfactorily to reduce dust contamination to the intake air before it travels further to inbye mine workings.

The improvements in air quality achieved through the installation of this innovative dust suppression system have significantly improved the quality of mine ventilation air to be delivered to underground mine workers and helped the mine continue its ongoing program of improving mine environment, health, safety and eventually mine productivity.

Figure 9.22 Impact of the new system on the respirable dust concentration along the roadway

Figure 9.23 Respirable dust reduction efficiency along the roadway
9.7 Summary

Dust particles generated from an underground bin are causing serious contaminations of the intake airflow to be supplied to inbye underground workings. Because of the complexity of the infrastructure and equipment movement around the bin, it has been impossible to gain a better understanding of the airflow and dust dispersion patterns using conventional dust monitoring devices. CFD modelling has been conducted to investigate both airflow and respirable dispersion patterns over the bin and along the belt roadway for the design of a better dust mitigation system. Modelling results show the dispersion of dust particles from the underground bin is mostly dictated by the ventilation airflow pattern, and will distribute widely in the belt roadway once they become airborne and appear at various elevations above the floor, contributing to high dust contamination of intake air in the belt roadway.

CFD modelling results show that ventilation from the horizontal intake at a rate of 10 to 13 m$^3$/s can be adopted to help dilute and confine the majority of dust particles from the bin downwards to the floor below the workers’ normal breathing zone. An innovative dust mitigation system based on the water mist technology was also proposed and subsequently mounted on top of the bin. The feasibility of this new system on respirable dust control was first investigated and verified from a theoretical perspective, after which a detailed design was approved for field implementation. CFD models were used to optimise the operating conditions of the nozzles in terms of spray direction. An independent field survey was then conducted to evaluate the effect of the new system on respirable dust capture, demonstrating a maximum of 68% of respirable dust reduction efficiency above the underground bin. It can be concluded from this study that:

- CFD modelling technique can be used to investigate the complexity of site specific airflow field in the underground ventilation system, especially when ventilation survey is not possible due to site constraints;
- Detailed respirable dust concentration dispersion can be predicted through CFD modelling studies;
Parametric studies can be conveniently conducted using CFD models to optimise the ventilation system for desired application, i.e. respirable dust dilution and hazardous gas dispersion;

Water mist technology has been demonstrated to be effective on respirable dust particles from the perspective of both theoretical analysis and field applications; the innovative dust suppression system can be applied to dust mitigation at other underground working locations, such as the longwall faces and development headings.
10 CONCLUSIONS AND RECOMMENDATIONS

10.1 Conclusions

Throughout the investigations conducted in the thesis, the capability of CFD modelling technique in solving the gas and dust related issues have been demonstrated. Specifically, the following conclusions were drawn:

10.1.1 Airflow and methane migration patterns on longwall faces

For the first time, longwall CFD models were developed close to actual longwall face geometry by incorporating the key features of longwall equipment, for instance, the belt conveyor and BSL in the MG, the chocks along the entire face, AFC, spill plate and the shearer with drums. An immediate goaf behind longwall chocks was added to the model as well considering the exchange of flow between face and goaf. Furthermore, the shearer position and its cutting sequence were taken into account to investigate their impact on the ventilation and methane flow behaviour along the face. Whilst more parameters were studied to understand the methane flow characteristics on longwall faces, including the flow rate, coal seam gas content, adjacent gas bearing strata, ventilation scheme and the drum sprays. The main findings obtained from the model results can be summarised as follows:

- Flow separation occurs at both the MG and TG inner corner where the flow boundary changes sharply, leading to the recirculation of airflow and subsequently the accumulation of methane gas at the two corners, therefore, special attention should be payed to monitor the methane concentration at these positions from a perspective of gas management at longwall faces;
- There is significant air leakage to the goaf as airflow enters the face from the MG, and the use of MG curtain would be able to effectively mitigate the air ingression into the goaf, so it is highly recommended;
• The source of methane causing accumulation at the tail end of longwall face could be identified, i.e., The majority of methane trapped in the inner TG corner comes from the longwall face while the methane accumulated at the upper TG corner is mainly attributed to the goaf methane emission;
• The rotation of drums affects the airflow patterns around the drums and thus helps the dilution of methane in its vicinity, especially when the drum sprays are in operation;
• Irrespective of the cutting sequence, the methane concentration at the TG drum is higher than it at the MG drum, and is the highest along the face;
• Methane emitted from the goaf is likely to accumulate at the upper TG corner where the airflow is approximately stagnant, and an additional TG cut through behind face would be efficient in diluting the methane at the upper TG corner;
• The methane distribution along the face is greatly affected by the coal seam geological conditions which determine the methane sources and the corresponding emission amount, ventilation system and the flow rate at the longwall face.

10.1.2 Gas flow characteristics in longwall goaf(s)

Numerical investigations on the gas flow characteristics in longwall goaf(s) were also carried out. Monitoring data obtained from tube bundle systems were adopted for the validation of base CFD models, through which good agreements between model results and monitoring data had been achieved.

For the gas flow in a single goaf, the spatial distribution of spontaneous combustion prone zones were identified and mapped, and various control strategies, which could be used to minimise the spontaneous heating zones, were assessed with particular attention to the optimisation of proactive inert gas injection strategies. Modelling results indicated that an optimum goaf inertisation could be achieved by pumping inert gas at least 100 m behind the face on the belt road side, or ideally via surface goaf hole(s). Goaf inertisation on the retaining wall side would only be effective for
localised heating, and should be used in combination with other control measures to minimise air dilution in these areas.

For the multi-goaf model, both the seam elevation and the position of the active longwall face were taken into account. Areas where spontaneous heating were most likely to occur were identified at different longwall retreating stages. Then parametric studies were carried out to investigate a number of operational scenarios and their impact on goaf gas migration behaviour. Observations obtained from the model results provided insightful solutions/guidelines to the management of gas flow in longwall goafs, in particular to the early detection of spontaneous heating and its control. Specifically, the results of multi-goaf models demonstrated that:

- Overall goaf gas flow patterns change as the operating longwall retreats owing to the changing ventilation pressure differential across the entire goaf areas as well as the varying goaf elevations along the longwall panels;
- Oxygen penetration into the active goaf remains high as the longwall retreats from the start-up position to finish-off line, reaching 15% or above even some 800 m behind the longwall face;
- If roof fall or roadway failure occurs in the perimeter road and restricts ventilation, there would be more serious air ingress into the goaf before the roof fall position;
- The most likely areas liable to the development of spontaneous combustion would be in the active goaf, the areas around the start-up seals of LW1-2 and LW4-5 and the adjacent goafs of LW7-8 at the middle part of the sealed areas;
- The gaseous products of spontaneous heating, such as CO and C₂H₆, behave similarly with most of the gaseous products dissipating into the goaf, gradually to appear along the seals of the perimeter roadway on the TG side of LW1;
- It would be difficult to detect an active goaf heating (on the MG side) at its earlier stage by solely depending upon CO/other gaseous products readings in return airflow, as the main stream of the gaseous product will be seeping out via seals along the perimeter road as well as dissipating into the sealed deep goafs in adjacent panels;
Dispersion of gaseous products such as CO for a heating is sensitive to the variation of face ventilation. High face ventilation will lead to extensive CO dispersion in the active goaf with diluted concentration towards the deep goaf and the perimeter road; whilst for the deep seated heating in the adjacent sealed goaf, CO dispersion is pushed further into the deep goaf, making it almost impossible to be detected by tube bundles or bag sampling around the goaf seals;

- Ventilation flushing of the perimeter road can help push the build up of high concentration methane into the deep goaf areas, however, this method is likely to cause excessive air leakage in certain parts of the sealed goaf;

- Goaf inertisation can be better achieved by pumping inert gas such as nitrogen at deeper positions (>200m) behind the operating longwall; the use of in-seam drainage methane for goaf inertisation will not be effective due to the buoyancy effect.

10.1.3 Respirable dust flow behaviour at longwall face and ventilation intake roadway above an underground bin

CFD studies were conducted to investigate the respirable dust flow behaviour for improved dust control at the longwall entrance and ventilation intake roadways, through which the dynamics of dust particles and its general dispersion patterns were obtained. It is found that the dispersion of dust particles was highly dependent on the flow field, and if not controlled properly, they would cause significant contamination to the ventilation system and propose threat to the health of mine operators. CFD simulations were also conducted to optimise the operating conditions of a new water mist generating system for the best dust mitigation effect at these working locations. Guided by the modelling results, field trials were carried out at two underground coal mines, and promising dust mitigation efficiency had been achieved. Specifically, field monitoring data had demonstrated that over 30% and 40% of respirable dust had been successfully captured at the longwall face and in the ventilation intake roadway respectively. The capability of CFD modelling in the successful design of site specific dust mitigation systems was demonstrated.
Therefore, throughout these studies, it can be concluded that the CFD modelling technique is an essential approach to gain fundamental understandings on the dynamics of fluid and particle flows, which are typically encountered in the ventilation system of underground coal mines. Furthermore, the understandings obtained from CFD models help the development of effective hazardous gas and dust control strategies, minimising the potential risks of safety and health issues which may be brought about during the practical mining operations.

Overall, from a perspective of CFD modelling study in underground coal mines, the highlights of the thesis can be summarised as:

- CFD modelling technique can be used to investigate the complex site specific airflow field in the underground ventilation system, i.e., the longwall face and underground bin. Validated by the main velocity profiles, a detailed airflow field predicted by the models is thought to be reliable, and it can be used to probe into the airflow patterns in some unreachable but critical zones, such as regions in the vicinity of drums and the immediate goaf, where field monitoring is not achievable using a conventional approach.

- Both gas and respirable dust flow characteristics in the underground environment can be investigated using the CFD modelling approach, which has been demonstrated by the validated application at the longwall face, in the goaf and above the bin. A significant characteristic of the approach is its powerful data visualisation technique, which provides better understandings of the spatial distribution and dispersion patterns of the hazardous gas and dust.

- Compared with implementing field tests, it is much safer and more cost effective to conduct parametric studies in terms of investigating the impact of ventilation and gas emission rate, optimising the inert gas injection strategies, and optimising the operating conditions of sprays. Aided by the data visualisation technique, the development and evaluation of effective control measures for hazardous gas and dust becomes easier and convenient.

- Depending on the purpose of research, the requirement for the degree of closeness between physical models and field conditions varies, e.g., the major
equipment is incorporated in the longwall CFD model to obtain a detailed airflow field and methane distribution patterns; the elevation change is taken into account for the multi-goaf models to investigate the goaf gas flow patterns without including the mining equipment along the active longwall.

10.2 Recommendations

It is recommended that further research be undertaken in the following areas:

- Full scale longwall face model with larger goaf area. For the longwall face model, it is necessary to extend the goaf length to 200 m or 300 m and the goaf height to 20 m or 30 m so as to better investigate the impact of different goaf compaction levels on goaf and face gas flow and distribution patterns;
- Transient modelling. Considering the impact of time factor, transient simulations can be conducted to gain understandings of the abnormal gas emissions along the face and from the immediate goaf, i.e., the occurrence of bumps ahead of face and periodic roof weighting in the goaf;
- Modelling coal oxidation in goaf. The prediction of spontaneous combustion prone zones can be greatly improved if the coal oxidation process is incorporated into the model, together with the temperature field;
- Development of TG shearer drum dust scrubber. Through the longwall face models developed in Chapter 4, the respirable dust dispersion patterns along the face and in the vicinity of the shearer can be investigated, meanwhile, field measurements and CFD parametric studies can be carried out to determine the optimum flow rate and position required for the TG drum scrubber, which could effectively reduce the dust levels and improve visibility downstream of the shearer along the longwall;
- Prediction of the mass concentration of respirable dust along the longwall face. Extensive field tests should be carried out to determine the respirable dust production rates from various operation processes, such as the drums cutting, AFC loading, BSL crushing and the belt conveyor. Thus, with the dust production rates correctly determined, a detailed mass concentration distribution of the respirable dust along the face is predictable using the
longwall models, which would be of great significance to understand the dust distribution along face and be beneficial to the development of better control strategies;

- Development headings and gas drilling stubs. Applications of the CFD modelling technique should be further extended to the development headings and gas drilling stubs, where the ventilation, and gas and dust management are of great significance to the safety and health of the underground workforces;

- Underground conditioning. Due to the progressive increase in mining depth and the intensively mechanised longwall faces and headings, the over-heating of ventilation flow has become a serious issue in some underground mines. Thus, the capability of the CFD models can be further explored to investigate the heat issue considering different heat sources in the underground environment, and can help in the development of corresponding heat control strategies.

- Ultra fine water mist. In addition to the use of ultra fine water mist system for dust suppression in underground coal mines, it has also been used as an effective fire extinguishment strategy in buildings and tunnels. The same as gas and dust hazard, fire hazard is another serious issue facing by many underground coal mines. Further studies can be conducted to investigate the feasibility of using this technique for fire suppression in underground coal mines. The ultra fine water mist system has a great potential for effectively suppressing the onset of dust and fire in underground workings.
REFERENCES


stress analysis." 30th International Conference of Safety in Mines Research Institutes. South African Institute of Mining and Metallurgy.


