International Journal of Mining, Reclamation and Environment

ISSN: 1748-0930 (Print) 1748-0949 (Online) Journal homepage: https://www.tandfonline.com/loi/nsme20

Computational fluid dynamics applied to mining engineering: a review

Guang Xu, Kray D. Luxbacher, Saad Ragab, Jialin Xu & Xuhan Ding

To cite this article: Guang Xu, Kray D. Luxbacher, Saad Ragab, Jialin Xu & Xuhan Ding (2017) Computational fluid dynamics applied to mining engineering: a review, International Journal of Mining, Reclamation and Environment, 31:4, 251-275, DOI: 10.1080/17480930.2016.1138570

To link to this article: https://doi.org/10.1080/17480930.2016.1138570

Published online: 22 Jan 2016.

INTERNATIONAL JOURNAL OF MINING, RECLAMATION AND ENVIRONMENT, 2017
VOL. 31, NO. 4, 251–275
http://dx.doi.org/10.1080/17480930.2016.1138570

Computational fluid dynamics applied to mining engineering: a review

Guang Xu[a], Kray D. Luxbacher[b], Saad Ragab[c], Jialin Xu[d] and Xuhan Ding[a]

[a]Department of Mining Engineering and Metallurgical Engineering, Western Australian School of Mines, Curtin University, Kalgoorlie, Australia; [b]Department of Mining and Minerals Engineering, School of Engineering, Virginia Tech, Blacksburg, VA, USA; [c]Department of Engineering Science and Mechanics, School of Engineering, Virginia Tech, Blacksburg, VA, USA; [d]State Key Laboratory of Coal Resources and Mine Safety, School of Mines, China University of Mining and Technology, Xuzhou, China

ABSTRACT
This paper provides a review of computational fluid dynamics (CFD) applications in mining engineering, with particular focus on mine ventilation-related flow problems. The basic principles of CFD are reviewed and six turbulence models commonly used are discussed with some examples of their application and guidelines on choosing an appropriate turbulence model. General modelling procedures are also provided with particular emphasis on mesh independence study and CFD validation methods, which can further improve the accuracy of a model. CFD applications in mining engineering research and design areas are reviewed, which illustrate the success of CFD and highlight challenging issues. It is expected that more CFD research will be carried out to solve problems in mining engineering, and the potential benefits from the simulations are enormous if proper modelling procedures are followed and modern computational approaches are implemented.

ARTICLE HISTORY
Received 23 June 2015
Accepted 3 January 2016

KEYWORDS
CFD; mining engineering; mine ventilation; turbulent flow; mesh independence study

1. Introduction

The principles of fluid dynamics are widely applied to underground mine ventilation, such as methane control, fire development, explosion, dust movement, and ventilation efficiency. Understanding of the mechanisms of fluid motion is essential for solving problems specific to mining, especially ventilation-related safety and health issues. Due to the complexity of the phenomena involved in mine fluid dynamics problems, computational fluid dynamics (CFD) modelling has been increasingly applied to the mining industry in recent years to accurately predict flow patterns, study flow mechanisms and results, and design equipment to improve the efficiency and safety of the mine industry. CFD is especially useful when a comprehensive analysis using physical experimentation requires expensive equipment, large amounts of time and understanding of flow in inaccessible areas.

CFD is a tool with which one can carry out numerical experiments for the purpose of determining indices that are impossible, or at least very difficult, to obtain from laboratory or full-scale experiments. The numerical experiments can not only be used to help interpret physical experiments, but also to better understand phenomena that are observed during physical experimentation [1]. As the

CONTACT Guang Xu Guang.Xu@curtin.edu.au

computational cost of CFD is dropping as a result of increasing speed of computers, and the cost of physical experiments is generally increasing, the amount of physical experimentation can be reduced considerably with the use of CFD. Not only can CFD be used to conduct virtual experiments, but it can also be used to better design physical experiments and increase efficiency.

CFD plays an important role as a research and design tool, and is a well-established technique applied to a broad range of fields including aircraft, turbomachinery, automobile and ship design, meteorology, oceanography, astrophysics, biology, oil recovery, civil, and architecture. Many of today's mining challenges require both analysis and visualisation of fluid flow behaviour in complex geometric domains, and CFD is a viable tool in that regard.

Because of the success of CFD, there are many publications of CFD studies in mining; however, very few include a review detailing the current state-of-the-art in mining research and development. The purpose of this paper is to present a review which provides current state-of-the-art information about the progress in CFD application in mining and illustrates its capabilities by way of examples. The emphasis is on general-purpose commercial CFD code methodology rather than specialised CFD software development.

2. Principles of CFD

CFD is one of the branches of fluid mechanics. It started in the early 1970s and employed physics, numerical mathematics and computer sciences to simulate fluid flows. CFD deals with numerical solution of differential equations governing the physics of fluid flow and the interaction of the fluid with solid bodies [2]. It uses numerical methods and algorithms to solve and analyse fluid flow problems. Flows of gases and liquids, heat- and mass-transfer, moving bodies, multiphase physics, chemical reactions, fluid-structure interactions and acoustics can be simulated through computer modelling [3]. The technique enables the user to predict what will happen under a given set of circumstances. The following section will give a brief introduction of the governing equations, along with the general methodology used in CFD.

2.1. Governing equations

CFD is based on the fundamental governing equations which express the fundamental physical principles of fluid dynamics: the conservation of mass, momentum and energy. Equations (1)–(4) show the conservation form of the partial differential governing equations, which are known as Navier–Stokes Equations [1].

The conservation of mass (the continuity equation):

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot \rho \vec{v} = 0 \tag{1}$$

where ρ is the density of fluid (kg/m^3); t is time (seconds); $\vec{v}$ is velocity vector (m/s).

The conservation of momentum (Newton's second law):

$$\frac{\partial (\rho \vec{v})}{\partial t} + \vec{\nabla} \cdot \left(\rho \vec{v} \vec{v}\right) = \vec{\nabla} \cdot p + \vec{\nabla} \cdot \vec{\vec{\tau}} + \rho \vec{b} \tag{2}$$

where $\vec{\vec{\tau}}$ is viscous stress tensor (Newton) given by Equation (3) below for a Newtonian Fluid, $\vec{b}$ is body force, and μ is the molecular viscosity coefficient.

$$\vec{\vec{\tau}} = \mu\left(\vec{\nabla}\vec{v} + \left(\vec{\nabla}\vec{v}\right)^T\right) - \frac{2}{3}\mu\left(\vec{\nabla}\vec{v}\right)\vec{\vec{I}} \tag{3}$$

The conservation of energy (the first law of thermodynamics):

$$\frac{\partial \rho e}{\partial t} + \vec{\nabla} \cdot \left(\rho e \vec{v}\right) = \rho \dot{q} + \vec{\nabla} \cdot \left(k \vec{\nabla} T\right) - \vec{\nabla} \cdot \left(p \vec{v}\right) + \vec{\nabla} \cdot \left(\vec{\vec{\tau}} \cdot \vec{v}\right) + \rho \vec{b} \cdot \vec{v} \tag{4}$$

where $\dot{q}$ is the rate of volumetric heat addition per unit mass, T is temperature, and e is internal energy per unit mass.

2.2. Turbulent flow

Reynolds number (Re) is a good indicator whether the flow is laminar or turbulent. It is calculated using Equation (5), for internal flow where ρ is the density (kg/m^3), u_{avg} is the mean velocity over the cross section (m^2/s), l_d is the pipe diameter or the hydraulic diameter for non-circular ducts (m), and μ is the dynamic viscosity of the fluid (Pa s). Under normal engineering conditions, flow through pipes at Re < 2000 may be regarded as laminar, Re > 4000 may be taken as turbulent flow and 2000 < Re < 4000 are treated as transitional flow, which is a mix of laminar and turbulent flow [4].

$$\text{Re} = \frac{\rho u_{\text{avg}} l_d}{\mu} \tag{5}$$

The flow in the mine gob is treated as laminar flow in porous media in many studies. However, the real flow inside the gob is still not fully understood, and the laminar flow assumption may not be valid [5, 6]. For most CFD codes, the modelling of transitional flow is usually not provided. But since most cases, the transitional flow only covers a small region of the total flow domain, it could be neglected and still acceptable results could be obtained [7, 8].

In underground mine ventilation, most flow state in mine openings are turbulent flow, which can effectively disperse and remove contaminants in the workplaces [9]. A large variety of turbulence models have been developed to solve turbulent flow problems, and some commonly used models are discussed below.

2.2.1. The standard k–ε model

The standard k–$\boldsymbol{\varepsilon}$ model is a two-equation model that computes the Reynolds stresses by solving two additional transport equations, which are for the turbulence kinetic energy, k in Equation (6), and the dissipation rate of turbulence, $\boldsymbol{\varepsilon}$ in Equation (7) [10, 11].

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho k u_i\right) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G_k - \rho\varepsilon \tag{6}$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho \varepsilon u_i\right) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{7}$$

where μ_t and G_k are given by:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \tag{8}$$

$$G_k = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_j}{\partial x_i} \tag{9}$$

The model coefficients are given by:

$$\left(C_\mu, C_{1\varepsilon}, C_{2\varepsilon}, \sigma_k, \sigma_\varepsilon\right) = (0.09, 1.44, 1.92, 1.0, 1.3) \tag{10}$$

The standard k–ε model is the simplest complete turbulence model and widely used in the modelling of mining turbulent flow. Yuan used the standard k–ε model in the ventilation airways to study the flow path in the gobs [5], and the spontaneous heating behaviour in the gobs [12–15]. Similarly, Ren and Balusu [6] used the standard two equation k–ε model to estimate the turbulent transport in his gob spontaneous combustion study. However, results of those studies were not validated. Collecutt et al. [16] used the standard k–ε model in the coal dust explosions study and the results are validated with test data from a coal dust explosion test facility. Toraño et al. [17] used the standard k–ε model and shear–stress transport (SST) model to evaluate the wind erosion effect to different coal pile. He eventually chose the standard k–ε model with a certain wall roughness parameter due to its better agreement with the US EPA wind tunnel measurements. In another study, Toraño et al. [18] used six different turbulence models to simulate dust behaviour for auxiliary ventilation in mining roadways and compared their results with the field measurements data, and the standard k–ε model provided better results. Silvester [8] states that the standard k–ε model is a more general but computationally intensive method and is favoured in mine ventilation applications. This turbulence model was also successfully used in a CFD study of dust dispersion [19] and several mineral processing studies [20–23]. Many other examples in the literature also indicated that it can provide precise correlation between the measured and the simulated results [24–28]. However, the standard k–ε model has also been reported to produce inaccurate results under certain circumstances, especially for flows with rotation, curvature, strong swirl, three dimensionality and flows with strong streamline curvature [3, 29]. This is partially because the turbulent viscosity hypothesis is not valid in those circumstances and the ε equation has many empirical constants which have adverse effects on the predicted results [30].

2.2.2. The RNG k–ε model

The RNG k–ε model is an improvement on the standard k–ε model and it is derived from the statistical methods used in the field of renormalisation group (RNG) theory [31]. It is similar in form to the standard k–ε model but includes modifications in the dissipation equation to better describe flows in high strain regions, and a different equation is used for effective viscosity. Details related to this model are discussed in [32].

2.2.3. The realisable k–ε model

The realisable k–ε model shares the same turbulent kinetic energy equation as the standard k–ε model, but uses a variable C_μ as shown in Equation (11), instead of constant, to calculate the turbulent viscosity using Equation (8) [11].

$$C_\mu = \frac{1}{A_0 + A_s \frac{U^* k}{\varepsilon}} \tag{11}$$

where $A_0 = 4.04$, $A_s = \sqrt{6}\cos\emptyset$, $S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$, $U^* = \sqrt{S_{ij}S_{ij} + \Omega_{ij}\Omega_{ij}}$, $\emptyset = \frac{1}{3}\cos^{-1}\left(\sqrt{6}W\right)$, $W = \frac{S_{ij}S_{ji}S_{ki}}{\widetilde{S}}$, and $\widetilde{S} = \sqrt{S_{ij}S_{ij}}$.

Also, a new transport equation is used for the dissipation rate, which is shown in Equation (12) [11].

$$\rho\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + \rho c_1 S\varepsilon - \rho c_2\frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} + c_{1\varepsilon}\frac{\varepsilon}{k}c_{3\varepsilon}G_b \tag{12}$$

This model is better than other k–ε models for many applications, especially improved modelling of planar and round jets, boundary layers under strong adverse pressure gradients or separation, and rotation and recirculation flows [11,31].

2.2.4. Reynolds stress closure models

The Reynolds Stress Model (RSM) is a higher level, more elaborate model. The turbulent viscosity hypothesis is not needed in this model and individual Reynolds stresses $u_i u_j$ are directly computed from the model transport equations [30]. The advantage of RSM is that it introduced terms accounting for anisotropic effects into the stress transport equations, which is very important for flows with significant buoyancy, streamline curvature, swirl or strong circulation [33]. More details about this model can be found in Pope's book [30] or Durbin's study [34]. RSM can produce more realistic and rigorous solutions for complicate engineering flow, but it requires more execution time and computer memory, and good convergence may be difficult [3].

2.2.5. Spalart–Allmaras model

This model was developed by Spalart and Allmaras [35], which is a one equation model first used in aerodynamic applications. In this model, the turbulent viscosity ν_T is solved by a single-model transport equation. The model equation is provided below:

$$\frac{\bar{D}\nu_T}{\bar{D}t} = \nabla \cdot \left(\frac{\nu_T}{\sigma_\nu} \nabla \nu_T \right) + \boldsymbol{S}_\nu \tag{13}$$

where $\boldsymbol{S}_\nu$, the source term, depends on the viscosity ν, turbulent viscosity ν_T, the mean vorticity $\boldsymbol{\Omega}$, the turbulent viscosity gradient $|\nabla \nu_T|$ and the distance to the nearest wall l_w [30]. This model is intended for aerodynamic flow, such as transonic flow over airfoils, and the application to the aerodynamic flows has proved successful [30]. For more details about the model, one can refer to the original publication [35].

Wala et al. [36] used different turbulent models to simulate the air flow and methane distribution in a ventilation test gallery. The results from the SST and the Spalart–Allmaras (SA) model were presented. Both models were successful in predicting the methane concentration and the airflow distributions, while the SST model is better in the box-cut scenario and SA model is better in the slab-cut scenario. Parra et al. [37] also applied the SA turbulent model in the study of deep mine ventilation efficiency, and found that good velocity agreement was achieved when compared to the experimental values. Sasmito et al. [38] compared four turbulence models, including the Spalart Almaras, standard k–ε, k–ω, and RSM. The results were validated by comparing with flow measurements from Parra et al. [37]. It was found the SA model achieved the best agreement with less than 15% relative error, and the computational cost is the lowest amongst others.

2.2.6. Large eddy simulation

Large eddy simulation (LES) directly represents the larger three-dimensional unsteady turbulent motions and models the effects of smaller scale motions [30]. A filter operation is applied to the Navier–Stokes equations to eliminate small scales of the solution. LES resolves large scales of the flow field and can be expected to be more accurate and reliable than alternative approaches such as RSM and RANS. It is much better suited to unsteady effects than RANS [39]. The computational expense lies between RSM and DNS models [30]. It is computationally expensive method that does not necessarily result in improved prediction results for fully developed turbulent flow, compared with the k–$\boldsymbol{\varepsilon}$ model [40]. Because it is at a much earlier stage of development than RANS modelling, few applications were found in the mining related fields. A CFD code developed by National Institute of Standards and Technology called Fire Dynamics Simulator (FDS) uses LES method and is commonly used to study mine fire. Some FDS mine fire studies are reviewed in Section 5.3.

2.2.7. Conclusion

There is no clearly superior turbulence model which works well over different applications. For general engineering turbulence modelling, Bakker recommends starting with the standard k–$\boldsymbol{\varepsilon}$ model [11]. For very simple flows that contain no swirl or separation, the calculation should converge with second-order upwind and k–$\boldsymbol{\varepsilon}$ model. For flow involving jets, separation or moderate swirl, the realisable

k–ε model and second-order difference scheme is most appropriate. If swirl dominates the flow, use RSM and a second-order differencing scheme. Other models should be used only if there is evidence from the literature that they are especially suitable for the problem of interest [11].

2.3. Numerical analysis

All methods in CFD use some form of discretisation which can be classified as finite-difference, finite-volume and finite-element. CFD can be approached using any of the three main types of discretisation mentioned above [41].

Finite difference method (FDM) is among the first approaches applied to the numerical solution of differential equations and is widely employed in CFD. It is applied to the differential form of the governing equations. It uses Taylor series expansion for the discretisation of the derivatives of the flow variables. FDM is simple and high-order approximations are available to achieve high-order accuracy. However, the application is restricted because this method requires structured grids and can only be applied to simple geometries. Thus, the finite difference methodology is rarely used for industrial applications [2].

Finite volume method (FVM), which is derived from the FDM, directly satisfies the integral form of the conservation law and uses the integral form of the governing equations. It discretises the governing equations by dividing the domain of interest into several arbitrary polyhedral control volumes, and satisfy the conservation equations on each control volume. FVM have two primary advantages which make its popular in commercial CFD packages, such as CFX, FLUENT and PHOENICS. The primary advantage is that the spatial discretisation is accomplished directly in the physical space. It naturally achieves the coordinate system transformation between the physical and computational domain. Secondly, FVM not only can be easily implemented on structured grids, but also do not require a coordinate transformation in order to be applied on unstructured grids. Therefore, the flexibility of FVM are particularly suitable for treating complex geometries [2, 42].

The finite element methods need the governing equations to transform from differential form to integral form and start with dividing the physical space into elements. These methods have the benefit of using the integral form and unstructured grids, which are preferable for complex geometries. However, they require much higher numerical effort compared to FVM, which makes it less popular [2].

3. CFD commercial software analysis process

Tremendous progress has been made in the development of CFD codes since the 1990s and the use of commercial CFD codes has increased dramatically in the last few years. The commercial CFD codes are the primary source of tools in use by the mining industry and other engineering communities. The powerful application of these commercial codes to model complex flow in many research and design fields makes them much more attractive.

There are generally three stages to perform CFD analysis: preprocessing, solving and postprocessing. Preprocessing is the first step in building and analysing a CFD model, taking place before the numerical solution process. The first step is to create the geometry of the problem. CAD geometries can be imported and adapted for CFD software. Approximations and simplifications of the geometry may be needed to analyse the problem with reasonable effort. Then a suitable computational mesh needs to be created and applied to the problem domain. After the mesh has been developed, boundary conditions and initial conditions should be specified according to the physical conditions. Finally, the flow problem is specified by the fluid parameters, physical properties and solving techniques.

A numerical method is then used to solve the discretised equations. Iterative methods are usually used until a predetermined convergence and required accuracy are obtained.

Postprocessing is the final step in CFD analysis. The results can be analysed both numerically and graphically. Some powerful commercial CFD software not only create visualisation graphs, including contour, vector, line plots and even animations, but also allow for data export to third-party postprocessors and visualisation tools, such as TechPlot. The visual presentation of the results allows the designer or researcher to interpret the data and have increased understanding of the flow mechanisms, and thus, better understand how the system responds to a variety of different operating conditions [43].

Although commercial software are usually user-friendly, the simulation process, especially analysing the results, requires complete understanding of the underlying physics, and often reasonable assumptions and improved boundary conditions are required to make the model manageable. Therefore, dependable results cannot be achieved without specialised training and sound engineering skills [43].

4. Procedures in CFD study

CFD is increasingly used in the research and design of ventilation and other fluid systems within the mining industry. Careful execution during the process of CFD studies is of paramount importance to make sure the high quality of the CFD results because modelling and numerical errors and large deviations may occur in such studies. Certain measures need to be taken in order to get high quality CFD results. Figure 1 shows generic basic steps in a complete CFD study. This section focuses on the discussion of two major steps: the mesh independence study and CFD model validation.

4.1. Mesh independence study

One important procedure emphasised by the authors is the mesh independence study. It is important to conduct the mesh independence study before accepting the CFD results since the numerical solution may depend on the mesh size if mesh independence is not reached [44]. By comparing the results of different mesh sizes, mesh independence should be studied considering different flow features at different representative locations. Flow features usually used to check mesh independence are velocity profile along a line, velocity contours and vectors at critical locations, temperature distribution, contaminant concentration, and other important flow features in the problem. One example can be found in the work of Xu et al., in which the distribution of a tracer gas under different ventilation scenarios was simulated using a conceptual mine model, and illustrated that the ventilation system can be remotely analysed using the developed tracer gas and CFD method [45]. Three meshes – coarse, medium and fine mesh – were built with the number of nodes progressively doubled. The results from these meshes were compared using point velocity and line velocity profiles. The results from the coarse mesh have larger deviations from results of the medium and fine meshes, but the difference between medium and fine mesh results was very small; the velocity difference at the monitor point is less than 2%. This indicated that the medium mesh is appropriate and sufficient for a robust solution, because the solution is relatively independent of mesh size. Fine mesh can usually provide a more precise solution, but once relative independence is met, it is a trade-off between gains in precision and increase in computational time. In studies by Diego et al. [46, 47] that compared the pressure drop modelled by CFD with that calculated by empirical and theoretical equations, they suggested that at least 12 tetras mesh cells should be used for a 5-m-radius tunnel, and the use of prisms meshes within the boundary layers close to the wall is necessary for accurate pressure drop modelling.

4.2. CFD model validation

Validation deals with the physical correctness of the CFD model. It is usually conducted by comparing simulated data to experimental data. This procedure provides the researchers and engineers the degree of confidence necessary for the CFD results application. This section discusses different techniques that have been used in the validation of mining CFD models.

Laboratory studies have traditionally been used to study mine ventilation related problems, and to validate the corresponding CFD models. The laboratory studies usually use physical scale models. In this case, the laboratory results are only valid when geometric and dynamic similarities are achieved

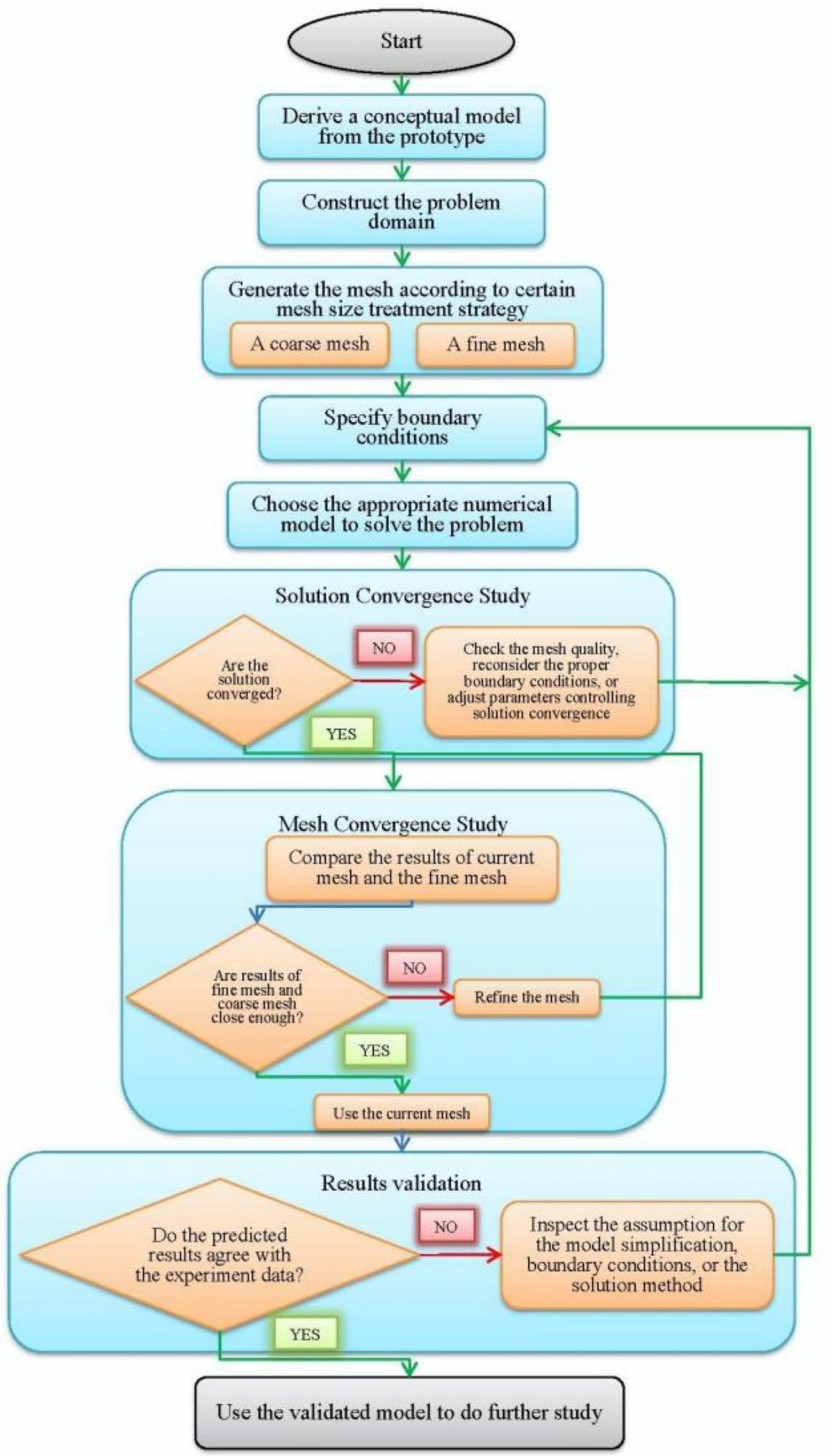

Figure 1. Basic steps flow chart in CFD study.

between the physical model and its prototype. Geometric similarity requires the model to have the same linear scale ratio in three dimensions, and dynamic similarity requires the model and the prototype have the same non-dimensional similarity parameters. For incompressible flow assumptions with no free surface, this requires the Reynolds number to be the same; and if a free surface exists, the Reynolds (Re), Froude (Fr) and Weber (We) numbers need to be the same. Compressible flow requires the Mach number to be the same [29]. However, dynamic similarity is often difficult or impossible to obtain. Moloney et al. [29] built a 1/10th scale model for the purpose of validating a CFD model of drivage face ventilation, but the model could not meet dynamic similarity, for it required 62 m/s exit velocity, which is not practical in the lab. The same challenge was faced by Ndenguma [26] in that a 15% scale model required an impractical exit air velocity of 152 m/s. Instead of meeting the dynamic similarity, a percentage volume flow method was used to scale the air flow.

Experimental results from the literature can also be used to validate a CFD model. In Toraño's CFD study [17], which is focused on the wind erosion of different coal stockpile geometries, the numerical model was validated by the US EPA experimental reference study to ensure the CFD model was accurate and valid. The root mean square deviation, shown in Equation (14), was used to quantify the difference between the CFD and the EPA experimental results, and a 3.75% deviation was found, indicating the CFD model is acceptable for engineering applications.

$$\mathrm{RMS} = \sqrt{\frac{1}{N}\sum_{i=1}^{N}(y_i - \hat{y}_i)^2} \tag{14}$$

Another method commonly used to validate the CFD model is to use data obtained from the field or an on-site full-scale experiment. For example, Peng and Xia [48] developed a 2D dense-medium separator CFD model to analyse the flow patterns and the mechanisms of particle separation in the separator. The in-plant test results, which showed a close fit to the simulated results, were used for the CFD validation. Similar validation can be found in Peng's other work on CFD studies of mineral separation [49]. Toraño et al. [50] used a handheld anemometer and methane detector to obtain velocity and methane concentration data which were then used for the CFD model validation. The best CFD model was chosen based on the least experiment and simulated differences. Parra et al. [37] conducted a detail ventilation measurement in a real mine gallery using hot wire anemometer to validate the numerical model. The grid size and the algorithm used for the simulation achieved good agreement with the experimental data. The validated model was then used to simulate different combinations of blowing and exhaust ventilation methods. However, sometimes it is hard to conduct accurate field measurements, and it is common for the measured field data have an error up to 20% [51].

The use of tracer gases started in the 1950s in building ventilation systems [52]. Tracer gas techniques have been used in many situations where the standard ventilation survey methods are inadequate [53]. The applications of tracer gases in underground mines include analysing ventilation patterns, measuring air leak rates, evaluating dust control methods [54], and evaluating the ventilation system after an emergency [55, 56]. For this reason, tracer gas is sometimes used for CFD model validation. Konduri et al. used CO_2 as tracer gas in their field experiment to study the effectiveness of jet fan used for auxiliary ventilation, and the measured results were compared with the CFD simulated results [57]. Krog et al. [58] used CFD to study the airflow patterns around the longwall panels and used SF_6 as tracer gas to validate the CFD model.

Flow visualisation has long been used in fluid flow research, and is also useful for CFD validation. A visual comparison of the results from the experiments and the CFD calculation provides effective means of CFD validation [59]. There are three basic visualisation techniques: addition of foreign material to the fluid stream, optical techniques, and adding heat and energy. Moloney et al. [29] conducted experiments using a laser sheet flow visualisation technique, combined with the CFD studies to evaluate different ventilation methods. In another study, Wala et al. [60] utilised a Particle Image Velocimetry (PIV) system to visualise the airflow patterns in a physical scaled mine model and validate CFD models. PIV is an optical technique that can measure flow components in a plane. Highly reflective tracer particles need to be added to the flow field. The motion of the particles can be recorded by a camera when illuminates the tracer particles twice within one camera shot using laser light. The processed results can display velocity vectors of the flow field and potentially be used to compare CFD's velocity vectors in the same plane. The use of smoke can also be applied to CFD validation. Ndenguma [26] used smoke to visualise the flow patterns in a scale mine heading model ventilated by jet fan and scrubber. The flow patterns imaged by camera were similar to those presented from the CFD results.

Velocity and turbulence statistics agreement between experimental and CFD results is a major indicator of validated CFD model. Therefore, it is necessary to accurately measure flow velocities and turbulence quantities. Moloney et al. [29] conducted a 1/10th scaled mine auxiliary ventilated headings experiment to validate the corresponding CFD model. Laser Doppler Velocimetry (LDV), which can measure two components of velocity without disturbing the natural flow patterns, was used to measure flow velocities and validate CFD model. Hargreaves and Lowndes [61] used a multisensor unit and vortex shedding anemometer, which are intrinsically safe devices, to measure and record air speed in an underground coal mine. However, because the anemometer can only measure airflow perpendicular to the measurement gate, and airflow in the mine is usually three dimensional, the measurement of the airflow and data interpretation is difficult. Therefore, a multidirectional intrinsically safe anemometer is needed to adequately map the complicated flow patterns in mines. Taylor et al. [62] used ultrasonic anemometers to measure three dimensional air flow velocities in a simulated mine entry. The results of the ultrasonic anemometer can be used to generate the quantitative flow profile using vectors, which includes the direction and magnitude of flow. Those results were later used for the CFD validation purpose that compared the experimental and simulated results in detail [36]. In addition, the advanced experimental methods, such as LDV and PIV, have been widely used to give detailed information on the turbulent flow field in stirred vessels and validation for the CFD studies in the process industry [20, 63–66].

5. CFD applications in mining

CFD modelling has been used as a tool of research and design in the mining industry in many areas to assist in understanding and analysis of the mechanism of fluid or gas flow in order to improve efficiency, safety and health issues. This paper organises mining related CFD applications into the following categories for convenience, although these categories are often interrelated.

5.1. Mine ventilation airflow

CFD is widely applied to the study of ventilation to improve the quality, quantity and control of ventilation, which can further assist in the provision of improved gas, dust and climate control [61].

Wala and others [36] conducted a series of studies aimed at validating the CFD code by comparing the CFD results against mining-related benchmark experimental results. He pointed out that although significant studies have been conducted using full-scale field tests or scaled physical modelling to evaluate the face ventilation system performance and have improved the ventilation effectiveness, there are still some doubts on the results due to the complexity of the ventilation and the limitation of experimental methods. Traditional theoretical and experimental methods can get valuable results, but they are limited in completeness and accuracy. CFD is a promising tool which embraces a variety of technologies and can overcome the disadvantages mentioned above if the CFD solution has been validated. In one of their study [67], CFD 2000 was used to determine the optimum design of an upcast shaft and main fan ductwork arrangement. The CFD model was first validated by existing experimental data available in other literature with reasonable agreement. Then the authors designed four different shaft collar and shaft cover arrangements and simulated them with the validated CFD model. The optimal design was determined by analysing the simulated results which could reduce the power cost due to less pressure loss. This CFD validation study went further in a later study [36], in which several experiments were conducted in a scaled physical face-ventilation laboratory model and CFD models were developed and compared with the experimental results for validation purposes. Two turbulence models were used: SST model and SA model. Both models were shown to be accurate for simulation of methane concentration and airflow distributions. Furthermore, a CFD study was carried out to study the effect of scrubbers on face airflow and methane distribution during the box cut mining sequence [68]. The SST turbulence model was used in the CFD simulation. The methane concentration results were compared under four different scenarios between CFD and experimental results. The results showed that the scrubber improved the face ventilation. However, further study was needed to determine the reason for the difference between experimental and simulated data. Some other CFD studies carried out by the same group of authors were presented in [69], which highlighted the use CFD in studying the underground airflow and improving the health and safety of miners.

Jade and Sastry [70] used a laboratory experiment and CFD simulation study to investigate the shock loss at the 90° intersections of two-way splits and junctions. The shock loss coefficient (SLC) results showed that the CFD models are validated well with the experimental data. The SLC results were compared with previous literature studies and the authors determined that the literature underestimates the SLC by 50% or more for two-way junctions, and 20% for straight branch, thus, concluding that the widely accepted methodologies also significantly underestimate SLC of two-way splits and junctions. The study also conducted regression analysis and obtained various equations for the estimation of shock loss for two-way 90°-splits and junctions.

Zheng and Tein [71] studied Diesel Particular Matter (DPM) in an underground metal/non-metal mine using CFD. The study investigated the airflow and diesel exhaust propagation patterns. The study assumed that DPM movement can be represented by the air flow pattern since very a small fraction of it exists in the air. A model was built, which represents part of a mine with highly mechanised room-and-pillar mining operation. The main air flow was simulated with and without stoppings. The model showed that although the DPM conditions are much improved by stoppings, some places still require auxiliary ventilation for adequate dilution. These places are dead end headings, cross cuts, and downstream of the backfill block. In order to solve the problem, the study also evaluated the effectiveness of both blower and exhaust system to reduce DPM problems. The CFD model simulated a single heading with a loader and truck operating in the immediate face. The results showed that with the blower system the DPM is distributed in a smaller space than the exhausting system, but the loader driver in both systems would be working in a high DPM environment. Therefore, other strategies are needed to improve the situation. In another study, Kurnia et al. [72] used CFD to study various auxiliary ventilation strategies for the removal of hazardous gases from diesel emission in a coal mine development heading. It used the standard k–$\boldsymbol{\varepsilon}$ model, and the gas consumption and emission rate at the boundary were set as pre-calculated values, thus no chemical reaction was included in the model. It compared the oxygen and carbon dioxide concentration when placing the continuous miner's exhaust pipe on the bottom and top of the machine, changing the ventilation duct's end direction, and the use of blowing and exhaust ventilation duct. It concluded that placing the continuous miner's exhaust pipe on the top of the machine and turning the end of the ventilation duct to the side could enhance the control of hazardous gases. However, some of the assumptions in this study were not realistic. For example, the result suggested to use a blowing auxiliary ventilation system in a coal mine development for better hazardous control, but the commonly used system in the US and Australia is exhaust ventilation for much better dust control. Overall, these studies showed that CFD can be used to simulate the airflow patterns for the entire mine or part of it, and the ventilation efficiency and different ventilation measures can be evaluated.

Aminossadati [73] investigated the effects of brattice length on fluid flow behaviour in the crosscut regions. CFD-ACE (ESI Software) was used in his study and k–ε turbulence model was employed. The results were compared well with the results of FLUENT for cross-validation purpose. The study indicated that airflow into the crosscut region was improved due to the use of brattice.

Parameters such as velocity and contaminants concentration are broadly studied to evaluate the underground ventilation system. However, the more restrictive parameters, such as mean age of air, which are used for evaluating ventilation in public places, are not commonly used to evaluate underground ventilation efficiency. Parra et al. [37] points out that the mean age of air and local levels of pollutants' concentration in risk areas are better factors to examine the ventilation quality. He used a validated CFD model to evaluate the effectiveness of three ventilation systems in deep mines: exhaust, blowing and mixed, by analysing the dead zones and the local mean age of air. The SA turbulent model was used, along with Navier Stokes equations for a three-dimensional, steady state, incompressible and isothermal flow regime. Dead zones are regions where velocity is below the regulated minimum velocity, but the dead zone criterion does not take into account the flow recirculation. The local mean age of air, $\bar{\tau}_p$, is obtained by solving Equation (15), and low air mean age indicate fresh air. [37]:

$$u_i \frac{\partial \overline{\tau}_p}{\partial x_i} = \frac{\partial}{\partial x_i} \left\{ \left(\frac{\nu_t}{\sigma_t} + \frac{\nu}{\sigma} \right) \frac{\partial \overline{\tau}_p}{\partial x_i} \right\} + 1 \tag{15}$$

The global efficiency, as shown in Equation (16), is used to compare different ventilation systems. An efficiency value of 1 represents a perfect displacement flow, and a value of 0.5 represents a perfect mixing flow.

$$\varepsilon_{\mathrm{GE}} = \frac{\overline{\tau}_{p,\ \mathrm{exit}}}{2\overline{\tau}_{p,\ \mathrm{total}}} \tag{16}$$

The results of the study indicated that although using the dead zone and local mean age of air criteria provides similar results, the local mean age of air criteria is more advisable due to it includes recirculation.

Zhang et al. [74] used the same criterion, the dead zone and age of air, to study the effectiveness of the push-pull auxiliary ventilation system. CFD modelling was used as an approach to calculate the effective range of an exhausting duct, for which no theoretical or experimental equation is available to calculate it. The air age was calculated using Equation (17).

$$\frac{\partial}{\partial x_j} \left\{ \rho u_j \tau_p - \left(\mu + \frac{\mu_t}{\sigma_t} \right) \times \frac{\partial \tau_p}{\partial x_j} \right\} = \rho \tag{17}$$

By examining the percentage of dead zones and the mean age of air of four different models, it was concluded that once the forcing duct position is determined, there is an optimum position for the exhausting duct in order to achieve the best efficiency, and in this study is 5 m from the working face.

CFD was also used to evaluate the fan effectiveness. Konduri et al. [57] used a two-dimensional CFD model to simulate a jet fan for auxiliary ventilation and obtained similar results with the experiments. Ray et al. [75] used CFD to simulate the performance of vertically mounted jet fans in a ventilation shafts for a passenger rail overbuild. The simulated results were compared with the calculated jet fan thrust results, and they determined that CFD analysis can be used to confirm key assumptions for other calculation methods if the CFD models are validated. Panigrahi and Mishra [76] used 2-D CFD models to simulate six different airfoil sections and study the study the effect of the variation of angle of attack on the fan blade aerodynamic coefficient. A most efficient blade design for mine ventilation fans was selected based on the results of the CFD models.

Mining is a dynamic process and airflow changes are associated with advance and retreat and the subsequent changes in mine geometry. However, it is particularly difficult to model a time dependent mining step together with the airflow simultaneously using CFD. Hargreaves and Lowndes [61] used a series of steady-state computational models to represent the different stage of tunnel drivage during the cutting cycle in order to assess the effectiveness and ventilation flow patterns of the machine mounted scrubber auxiliary ventilation system. The cutting cycle was decomposed into several representative steady-state stages and 24 simulations were carried out to replicate the whole cutting cycle. The simulation results were compared with the full-scale ventilation experimental data. This study shows that CFD modelling can improve the understanding of auxiliary ventilation systems during different stages of cutting cycles and the results can be used to improve the planning and operation of auxiliary ventilation systems. The results of this study were later used as a data library integrated to virtual reality technology to develop an improved ventilation planning and training tool [8]. The dynamic-mesh (DM) technique also can be used to solve similar problems with moving mechanical components and changing domain shapes. Using the DM method, the computational mesh is not static, but is dynamic with changing shape and structure to simulate a dynamic process. The DM method can be applied to the simulation of many dynamic mining flow problems, but few applications were found in mine ventilation CFD modelling.

The tracer gas technique is a useful and versatile tool for studying mine ventilation systems. A research project conducted at Virginia Tech involved the selection of novel tracer gases for mine ventilation, the development of a methodology to use tracer gases and CFD modelling to analyse, predict and confirm the underground ventilation status together with the location of the damage, and finally validate the developed methodology in the laboratory and in the field. The outcomes of the project were published in several forums [45, 55, 56, 77–85], in which the CFD modelling proved to be able to assist with tracer gas experimental design for reducing the trial-and-error processes expended in the field. The modelling results can provide information for determining some of the key experimental parameters, such as the release rate and duration, the expected concentration profile, and the release location. After these parameters are optimised in the CFD model, the trial-and-error process can be reduced and the desired results can be obtained more efficiently. It also proved that the use of CFD models and tracer gas techniques can successfully be used to analyse underground ventilation systems after and emergency event.

5.2. Spontaneous combustion

Spontaneous combustion often occurs in the gob areas where it is difficult to detect, locate and extinguishment [5]. However, it is difficult and expensive to carry out large-scale experiments to study spontaneous heating in underground mines [86].

Yuan et al. conducted a series of large-scale CFD numerical modelling studies on spontaneous heating. He studied spontaneous heating in typical long wall gob areas with bleeder and bleederless ventilation systems with a stationary longwall face [12, 15, 87]. The estimated gob permeability and porosity profiles from a geotechnical model were used as inputs for the CFD model using Fast Lagrangian Analysis of Contoinua (FLAC). The Kozeny–Carman equation was used to estimate the changes in permeability in the caved rock. The flow in the gob area is treated as laminar flow while fully turbulent flow was applied to ventilation airways. The studies showed the flow patterns inside the gob, and the effect of other critical factors that influence spontaneous combustion. These studies proved that CFD is an effective tool for the study of spontaneous combustion, and the modelled results were reasonable compared to previous experiments and studies.

The effect of barometric pressure changes on spontaneous heating in longwall panels was presented in another article by Yuan and Smith [13]. The actual recorded barometric pressure variations were used in a bleederless ventilation model and the oxygen concentrations were quantitatively examined. Results showed that the barometric pressure change has slight influence on the maximum temperature of the spontaneous heating in the gob, which is also affected by the gob permeability and the coal oxidation rate.

Another study by Yuan and Smith examined spontaneous heating in a coal chamber utilising CFD [86]. The results were validated by comparing results with a test from U.S. Bureau of Mines and achieved similar phenomena. The study results demonstrate that the CFD model has the ability to reasonably reproduce the major characteristics of spontaneous heating, the results are in good agreement with the experimental results and the model is useful for predicting the induction time, which is very important to prevent spontaneous heating fires.

5.3. Mine fire

Mine fires are another challenging underground mine safety issue. The toxic gas and low visibility caused by fire creates an extremely hazardous environment in underground. Miners can be seriously injured by inhaling toxic products-of-combustion (POC), and the fire heat can cause rib and roof collapse [88, 89]. Underground coal fires also produce large amounts of CO_2; for example, in one study in China, nearly 100–200 million tonnes of coal affected by underground coal fires were calculated to produce 2–3% of the total world CO_2 emission [90]. A number of studies are related to the investigation of mine fire and its combustion products using CFD. It was pointed out that the CFD-based tunnel fire models represent the best predictive modelling because CFD makes few assumptions compare to other methods [91].

A CFD study was conducted by National Institute for Occupational Safety and Health (NIOSH) and Mine Safety and Health Administration (MSHA) to investigate the temperature characteristics of mine fire [92]. Two deep-seated fire tests were conducted: one fire consisted of coal material and the other consisted of a mixture of coal and wooden cribbing blocks. The model was built using FDS, which is a CFD programme developed by National Institute of Standards and Technology. Due to the complicated geometry for the mixed fuel fire scenario, only the fire consisted of coal material scenario was modelled. Some actual observed phenomena could not be duplicated in the model due to the fact that coal was treated as a continuous medium in the model. The simulated maximum surface temperatures were very close to the measured results. However, the model overestimated the average flame spread rate, which may be also caused by the continuous medium treatment of coal in the model, while in the experiment it consisted of packed pieces of coal.

Edwards et al. conducted a series of CFD study to understand mine fire and smoke spread. In one study [93], a CFD model was built using CFD2000 to model buoyancy induced POC spread from experimental fires in the laboratory and to analyse smoke flow reversal conditions. For fires in an entry under zero airflow condition, the predicted spread speeds were higher than the measured values, which were explained as the POC was not distinguished from airflow in the model. The predicted temperature was also higher than the measured value, which was due to the high transport velocity and the negated heat loss at the roof. Based on a computation using C_3H_8 as fuel, the CO concentration results agreed with the measured data although with early alert time, which is a result of the sensor response time and the transient fire growth in the experiments. For fires in an entry under positive airflow condition, smoke flow reversal conditions were analysed. The CFD predicted critical velocity values were lower than that calculated by Froude model, which favourably predicted the measured smoke reversal. The study showed the limitation of CFD model with incomplete experimental conditions, but CFD can still provide useful results regarding to the parametric influence of experimental conditions. A similar study was presented in [94], in which fire in a tunnel was simulated by a diffusion flame of propane in CFD2000. The computed velocity and temperature profiles along tunnel cross sections were similar to available experimental results. It also correlated reverse-flow layer length with the Froude number which can provide guidance for smoke management.

Another study presented an experiment and computational model to determine the critical ventilation velocity required to prevent smoke reversal [89]. Fire smoke reversal experiments were conducted with different fire intensities and found that the critical velocity to prevent smoke reversal is proportional to the fire intensity to the 0.3 power which is in agreement with the one-third law dependence theory posted by other researchers [95]. The CFD model using FDS showed good agreement with the experimental results, which provided a predictive method to simulate a range of fire intensities and mine entry dimensions that is difficult to achieve experimentally. In another study, mine fire caused by different fire sources was modelled using FDS [88]. To study coal fire spread, a model was built to simulate the 1990 fire at Mathies Coal mine. Similar to previous studies, the predicted flame spread rate was also higher than the reported rate. This was stated due to the presence of inert materials in the mine rib and roof. The flame spread rate was found to be a linear function of airflow rate. To study the timber sets fire spread, another model was built based on the investigation of Warner [96]. The predicted flame propagation rate was also found higher than the measured results. A good agreement was achieved for the gas temperature prediction during the temperature increase and decrease period, but the maximum temperature was underestimated with the model. To study the conveyer fire spread, CFD simulations were made base on available experimental data. A good agreement was achieved considering the air speed influence on flame spread rate. Overall, the study showed CFD's capability in modelling mine fire caused by different fire sauces, which can potentially be used to help mien fire emergency planning.

Huang and others [90] presented a CFD method using a two-dimensional model which is based on the theory of natural convection and heat transfer in porous media to study the flow and temperature fields in underground coal fires. The solutions compared well with the limited available field data. The results showed that the fractures or high permeability are important factors in enhancement of natural convection. In a uniform permeable stratum, air flows from the low temperature zone to the hot area, but in a non-uniform permeable stratum, air flows from the more permeable zone to the hot area and less permeable zone. The study also found that air convection influence shallow coal seam fires more than deep coal seam fire, and the gas produced by secondary combustion in fractures can enhance the convection.

5.4. Methane flow and control

Methane in underground coal mining is one of the major safety issues. Methane is highly explosive under certain concentration and requires constant monitoring and control to maintain a safe working condition. The gas flow in the gob and mine ventilation system are complex. CFD can be used to better understand complex underground methane flow and design ventilation methods to reduce the methane risks [97].

Ren et al. [98] presented a CFD modelling study of methane flow around longwall coal faces. Due to the fact that methane in the working face is generally from source beds above or below the working seam, this study constructed a model that has a methane bearing seam 80 m above the working seam. Laboratory results were used for the permeability values of the roof strata, as well as the consideration of redistribution of stress field and the mining induced fractures. The pressure and velocity contours were provided, which related to the methane emission and migration. Although the CFD model provided practical results in their studies, validation is needed by comparison with field data. Kurnia et al. [99] proposed an intermittent auxiliary ventilation method for methane removal and cost-effective development heading ventilation. In order to save power cost, the air quantity provided to the heading was proposed to alternate between a predetermined high and low flow. However, there are two major issues with this method. The first is the control of such alternating flow is very challenging if practical at all. Secondly, one primary assumption is that the methane concentration is still below the regulated limit during the period of low flow. If this is the case, keeping the flow at this low flow rate would save more power cost than alternating low and high flow.

Toraño et al. [50] conducted CFD analyses of methane behaviour in underground coal mine auxiliary ventilation. The conventional method calculates the average methane concentration without considering different methane content in different zones. His study aims to analyse the evolution of ventilation in different cross sections and in the roadway axis directions, and the influence of time. A CFD model using CFX and field experiments were conducted to study the dead zone, airflow recirculation and methane distribution. The study compared four different turbulence models, which are SA, k–ω, k–ε and SST model. The k–ε model was finally selected because it agrees with the field measurement best. The study shows that it is necessary to analyse auxiliary ventilation systems by CFD which helps identify potentially dangerous zones and inform design of auxiliary ventilation.

Coal mine production may be prohibited by high methane content underground, especially when the ventilation is not sufficient to lower the content with normal ventilation systems. Oraee and Goodarzi [100] used CFD to simulate a methane drainage system which can be used to reduce the ventilation and development cost in a gassy mine. The study evaluated the methane drainage system with different vent hole spacing and the change of methane content with time. The study showed that the CFD model can be used to improve the drainage system design which will effectively manage methane underground. Balusu et al. [101] also presented an extensive study on the optimisation of gob methane drainage system. Several techniques were used during the course of the project, such as on-site monitoring, tracer gas tests, CFD simulations and field trials. The CFD method was used to analyse the gas flow and buoyancy mechanisms in the gob. The models were validated and calibrated using the field study results. The influence of different parameters, such as face flow rate, drainage hole position and spacing, is investigated. The CFD results, in combination with other field investigations, were used to develop optimum gob gas control strategies. The gas drainage strategies developed by this study demonstrated approximately 50% gas drainage improvement compared to the traditional gob gas drainage strategy which enhanced the safety and productivity of underground coal mines.

5.5. Gob gas flow

It is important to understand the mechanics of gas flow inside the gob in order to develop effective gas management and ventilation strategies. However, it is hard to measure the air flow inside the gob because much of the gob area is inaccessible. Therefore, the modelling technique is one reasonable way to investigate the ventilation in gob areas [12, 102].

Permeability distribution in the gob is a key element of the gob gas flow model. Esterhuizen and Karacan [103] developed a methodology for calculating permeability variations in the gob suitable for reservoir model or CFD models and simulated the leakage flow into the gob, methane distribution and effects of gob vent boreholes on flow patterns. The permeability changes were determined using FLAC3D numerical modelling programme and the results is used as input into the reservoir model. The simulated results are consistent with empirical experience and measurements reported in the literature.

A similar study was conducted by NIOSH [5]. The flow patterns inside the gob under one-entry and two-entry bleederless systems, and a three-entry bleeder system were studied using CFD. Gas flow in the gob is simulated as laminar flow through porous media. The gob permeability data were derived from the results of FLAC geotechnical modelling. The study also discussed the possible location of critical velocity zones which support spontaneous combustion. The studies of gob gas flow and its success are an essential step for spontaneous combustion modelling conducted by the same group of authors shown in Section 5.2. In another study by Tanguturi and Balusu [104], CFD models were used to investigate the high methane-level problems on the tail gate side of gob. The model with 1000 l/s methane emission rate resulted in high level methane concentration at the tail gate region with ventilation rate of 60 m^3/s to the longwall panel. In this situation, electrical equipment at the longwall will be tripped off and all the mining operations need to be ceased. It is challenging to manage such issues by using ventilation air alone. However, with strategies of gob drainage, back return ventilation system, and installation of curtains across the face, the high-level methane concentration hazards were eliminated.

5.6. Inertisation

The goal of gob inertisation is to lower the risk of spontaneous combustion. Gob inertisation has been widely used around the world to control fires and spontaneous heating in underground coal mines. The effectiveness of inertisation can suppress the development of potential gob heating and maintain a normal coal production rate [105]. High wall systems have also effectively utilised inert gas to maintain safe methane levels [106]. Studies have shown CFD can be used to better design inertisation.

Ren and Balusu et al. [105, 107] presented their work using CFD to study the optimum inertisation strategies which can achieve gob inertisation within a few hours of the sealing the panel. The CFD model was first calibrated based on previous inertisation studies and gob gas monitoring. The gob porosity parameter was determined from the results of geomechanics models. The validated model was used for parametric studies, including inert gas injection locations, inert gas flow rates, seam gradients and different inertisation strategies such as injection of inert gas through surface gob holes. Results showed that the inert gas composition is not the major factor in an inertisation process and that injection of inert gas at 200 m behind the face is more effective than at the location right behind the face line. Several recommendations were provided to improve the inertisation strategy and the strategy developed by this study was implemented and demonstrated in the field. The new practice showed promise in converting the gob into an inert atmosphere in a few hours instead of two to four days by traditional methods. The same group of authors [108] conducted another CFD study to assess the effects of various parameters and inertisation options for a blasting gallery coal mine panel, and the according field trials proved to be successful.

Mossad and others [106] studied the effectiveness of high wall mining inertisation using CFD. The study focused on improving the mine's efficiency with regard to safety and production rates by using inertisation to maintain methane concentrations within safe working limits. The model is a 2-D, k–ε realisable turbulent model. The study indicated that applying the inert gas at high angles is more effective and CO_2 is the most effective inert gas, when applied at a 60 ° angle, compare to N_2 and Boiler Gas. This work was described in more detail in Vella's dissertation [109].

Trevits and others [110] conducted CFD modelling using FDS to study the effects of the inert gas (N_2) injection rate. The results of the model achieved good match with the field test which used the pressure swing adsorption N_2 generation technology to inert a mine sealed area. The results showed that the relationship between N_2 gas injection rate and the time needed to reduce the O_2 level is not linear and the benefit of inert gas decreases as the injection rate increases. The CFD results also show that injection of N_2 at two ventilation seals is more efficient than at one seal location with a double injection rate.

5.7. Dust dispersion and control

The amount of dust generated during mining is another major concern. Dust can cause respiratory disease, contribute to the risk of underground explosion and impede productivity [111]. The airflow and dust dispersion are very complex and the standard mine ventilation network analysis is not sufficient to analyse the detail airflow patterns and dust distribution. CFD is an attractive approach to develop and evaluate dust control method.

Heerden and Sullivan [111] completed a CFD study to evaluate the dust suppression of a continuous miner and a roadheader. The study showed the steps during the CFD model constructions, and plotted the velocity vectors and contours of the results. The dust particles are assumed to follow the flow in the flow field, and the flow path lines were used for qualitative assessment of dust movement. The model was used to evaluate dust suppression under different machine parameters and dimensions, such as the position of the continuous miners, the volume of the flows, and different models of roadheaders. The effect of drum rotation, water sprays, and air movers were also investigated. The model is stated to have been validated by comparing with the experimental data, although no details were provided for the validation.

Srinivasa et al. [112] studied airflow and dust dispersion at a typical longwall face using CFD. The study evaluated the air curtains, semi-see-through curtain and air powered venturi scrubber dust control techniques. The effect of support legs and shearer were included in the model with simplified geometry. The dust is assumed inertialess and follows the air flow streams, which is assumed in most CFD dust studies. The dust was added in the model as a source at the model inlet and was assumed to be constant and uniform across the inlet. The flow field equations were solved independently from the pollutant equation. Detail equations used for solving the movement of dust can be found in the original paper. The air velocity results and the dust concentration values using air curtains were compared with field measurement. The predicted dust concentration was within 10% of the field values. The simulation indicated that the air powered venturi scrubber is the most effective means to control dust, with a 40–50% reduction within a distance of 3–4 m from the scrubber at 2.1 m/s face air velocity. The study concluded that CFD can be used to model underground dust dispersion and design dust control techniques.

Silvester et al. [27] presented a CFD study on the influence of underground mineral tipping operations on the surrounding ventilation system and consequent dispersal of fugitive dust. It used a two-phase continuum approach to describe the interaction between the falling materials and the surrounding air. The standard k–ε model was used and wall roughness effects were not considered since it was proved to have negligible impact on subsurface mine ventilation modelling [113]. Different scenarios were modelled to investigate the influence of different factors. A Lagrangian particle tracking algorithm was used to represent the dust flow and plume dispersion. The CFD results were validated against the experiments using scale models, which used water as a substitute for air to achieve adequate dynamic scaling, and used a dye injection system to visualise the flow. Good qualitative agreement was achieved between the experimental and the CFD results. However, the use of continuum dynamics to represent the material as a granular fluid medium restricted the CFD model so that it could only achieve an approximation to the actual process and could not reveal the mechanisms of the process. Another CFD study conducted by the same group on the dust control aspect was presented in [114]. The dispersion and deposition of fugitive mineral dust generated during the process of a surface quarry excavation were studied. The influence of the mineral dust emission location, the wind direction and the in-pit ventilation flows were investigated. Although no experimental data were provided to validate the CFD models, the results can potentially be used to assist future quarry planning and blast operation to better control the dust emissions.

Toraño et al. [17] used CFD to model different shapes of open storage systems for bulk materials, such as coal and iron ore, to study the best operational and investment parameters to reduce the airborne dust. US EPA established a methodology to estimate the level of airborne dust generated from an open pile. However, sometimes the existing methodology or standard do not match the way to store materials due to reasons like area restrictions and stacking means, and it is usually not easy to carry out experiments to evaluate the level of airborne dust to compare to the standards. Therefore, CFD was used to predict the different environmental impact of different storage piles. The model was validated by comparing results for cone and flat top oval piles with the US EPA study, and then a semicircular pile model was built and analysed using the validated model method. The model results showed good agreement with the US EPA study with low root mean square values. The semicircular pile model showed a lower emissions and wind erosions level, but the wind direction is an important factor that would influence the results.

Skjold et al. [25] reported a CFD code Dust Explosion Simulation Code (DESC) which is a simplified empirical-based CFD code that can be used to simulate the dust lifting phenomenon. The empirical approach was used for the DESC code since the detail dust lifting mechanisms, such as the Magnus forces, Saffman forces and particle collisions, cannot be feasibly modelled. The DESC code is similar with the CFD code Flame Acceleration Simulator (FLACS). The dust particles are assumed to be in dynamic and thermal equilibrium with the fluid phase. The phenomena such as dust settling or flow separation in bends and cyclones cannot be modelled due to the fact that slip velocity is not included in the code. The study simulated dust concentration for an experimental wind tunnel, as well as a set of dust explosion experiments described in literatures. Although the experiment's technical details were limited, the simulated dust layer results agreed quite well with the experimental data. As a result of this study, it concluded that it is beneficial for the safety of coal mines or other industrial field to use a simplified dust lifting model.

Ren and Balusu [115] presented their work using CFD to develop new dust control systems. The geometry of their models were comprehensive, including not only the coal face and the maingate, but also chocks, shearer, spill plate, BSL/crusher and conveyor, dust scrubbers, shearer clearer, venturi sprays, and curtains. One particular example they showed is the use of CFD modelling to design a new shearer scrubber system. By studying parameters such as the location of inlet and outlet, the capacity of the scrubber, and the face airflow rates, the study indicated that orienting the scrubber inlet towards face ventilation can capture more dust particles. The CFD modelling results were used for a new shearer dust system design, and achieved 43–56% dust reduction.

Torno et al. developed a CFD model to simulate the dispersion of dust generated during blasting in limestone quarries [19]. Similar to Silvester's study [27], the standard k–ε model was used for turbulence modelling and Lagrangian particle tracking method was selected to model the air and dust multiphase problem. The CFD model was validated by the experimental data using a trial-error method on the value of dust injection. It found that the use of a barrier placed downstream of blasting can create 4.5% of dust emission retention. The study illustrated better barriers can be designed, which can minimise dust pollution, by using CFD model that takes into account blasting characteristics, meteorological conditions and ground morphology.

Coal dust explosions are one of the most significant hazards in underground mines. Collecutt et al. [116, 117] did a series of comprehensive studies by using CFD simulations to model coal dust explosions and the effectiveness of active explosion barriers. The models were calibrated by the experiment results from the Siwek spherical 20 L chamber at Simtars and the Kloppersbos 200 m long explosion tunnel in South Africa. The models were later refined by accounting for the particle collision model and the spray breakup model, which yielded excellent agreement against the experimental data. These studies are one of the few that used the open source CFD programme 'OpenFOAM'. It allows for a wider range of user defined options and particle–particle interaction models, and the use of such programme is to be encouraged in the mining CFD applications.

5.8. Minerals processing

CFD has been extensively used in recent years in the process industry for the research and development of new and existing processes. CFD modelling results can help researchers to gain detailed understanding of flow during minerals processing that can be used to design and modify equipment to improve separation performance. Numerous studies were available in the literature devoted to the CFD modelling of minerals processing. For example, a study was presented by Lichter et al. [118], which used the combination of CFD and discrete element modelling (DEM) to evaluate the performance of flotation cells. CFD was used to simulate a flotation machine with different parameters, such as the size of the flotation cells and inlet velocities. Slurry was treated as single phase Newtonian fluid with specified viscosity. The model did not include the air in the slurry system, but the author states that it still can be used to compare one cell design with another. The CFD results were then imported to a DEM simulation, which makes it possible to produce residence time distributions as a function of size and evaluate the metallurgical performance. No final conclusion was made on the relationship between parameters and the performance of the cell. However, this study showed the potential of the combination use of CFD and DEM modelling to optimise the flotation cell operating and design parameters.

5.9. Other applications

Applications of CFD also have been found in the study of other aspect of mining engineering. Berkoe and Lane presented several projects which applied CFD to mining and metals field [43], including quench cooler design to reduce process gas temperature, solvent extraction settler design to achieve uniform flow, plume capture performance prediction for different configurations of fugitive emissions collection system, effect of wind on operations facilities study, performance of a ferro-nickel smelting furnace and slurry flow distributor design. They especially highlighted that the CFD study requires deep understanding of the underlying physics, and usually needs to apply simplified assumptions and improve boundary conditions. Therefore, it is important to have an interface person between the field engineering and the CFD modelling functions to obtain reliable results.

CFD has also used to study the water pollutant associated with mining. Ardejani et al. [119] used PHOENICS to study the acid mine drainage generation and subsequent pollutants transportation. The chemical reaction process was implemented to the model by subroutines. Close agreements were achieved between CFD model results and filed data. This study illustrated the ability of CFD to study the groundwater pollution problems and better understand the pollution transport mechanisms.

CFD has also been applied in the field of carbon capture and storage (CCS) technology. Mazzoldi et al. [28, 120] presented a risk assessments study for CO_2 transportation using a commercial CFD software Fluidyn-PANACHE. In their study, models were built to simulate an accidental release of carbon dioxide from high pressure transportation facilities within CCS projects. The results were compared with those using Gaussian/dense-gas models and demonstrated that CFD models are more reliable and precise, providing improved risk analyses. A similar study conducted by Dixon et al. [121] used CFX to predict the consequences of releases of carbon dioxide from a liquid inventory. The concentration of CO_2 particle was modelled using both scalar equation and Lagrangian particle tracking methods and good agreement with experiment has been observed.

6. Conclusions

The CFD basic concept and its application in the mining industry have been discussed in order to provide an insight into the current CFD research activities in mining. It is evident in this review that the scope and the level of sophistication of CFD study in mining is increasing, especially with the advancements in computer power. The application of CFD in the mining industry will benefit the understanding of the fluid problems that can improve the safety and optimise layout and equipment design.

Turbulence models are widely used since most flow in mining is turbulent flow. It is clear that the standard k–ε model has been commonly used as the most acceptable general purpose turbulence model. However, the quality of the solution is dependent on the turbulence model. Therefore, the selection of turbulence models should consider physical model and flow features in specific problems.

Mesh independence study is a required practice in the construction and analysis of a CFD model. This paper summarised the general procedures and methods to conduct the mesh independence study, which assure the solutions not change significantly with further mesh refinement. Mesh independence is an important step to guarantee a robust CFD modelling; therefore, it is a highly recommended procedure in CFD studies.

Because CFD uses approximate approaches and some assumptions, validation of the CFD study is necessary to make sure the simulated results are within an acceptable level of accuracy. The validation studies are generally conducted by comparing the results obtained from laboratory or full scale experiments with the simulated results. Several experimental techniques were used to produce data for CFD validation, such as tracer gas, 3D velocity measurement, and flow visualisation. Accurate measurements of flow features sometimes are extremely difficult due to the complexity of and access to the flow domain, such as underground mine working face and flow in the gob. General agreement with the experimental data was reported in many validation studies, whilst discrepancies were also reported in some studies, which requires model improvement and accurate measurements of flow parameters. More paramedic study and validation research should be encouraged in mining-related studies.

Some modern CFD approaches, such as adaptive meshes to save computational time and dynamic meshes for dynamic mining flow problems, are not discussed in detail in this paper, but they are beneficial to CFD researchers. Very few CFD studies related to mining have employed those methods. Most of the cited studies used commercial CFD codes. However, these codes have limitations, such as they sometime cannot adequately model multiphase flows of suspensions and mixtures, and they have limited selections of user-defined models. The more use of user defined models and open source CFD codes (such as OpenFOAM) need to be promoted in the future.

Overall this paper reviewed the current state of research of CFD modelling in mining and provided a good number of references for the convenience of mining engineering CFD researches. Examples discussed in this paper and numerous studies that can be found in the literature showed that the potential benefits from the CFD simulations are enormous if the problem setup was addressed carefully and proper model procedures, such as mesh and solution convergence, and model validation were conducted.

Acknowledgements

This publication was developed under Contract No. 200-2009-31933, awarded by the National Institute for Occupational Safety and Health (NIOSH). The findings and conclusions in this report are those of the authors and do not reflect the official policies of the Department of Health and Human Services; nor does mention of trade names, commercial practices or organisations imply endorsement by the US Government. The authors also would like to acknowledge the partial support from the Excellent Innovative Project fund from China University of Mining and Technology under project no. 2014ZY004.

Disclosure statement

No potential conflict of interest was reported by the authors.

Funding

This work was supported by the China University of Mining and Technology [2014ZY004]; National Institute for Occupational Safety and Health [200-2009-31933].

References

[1] J.D. Anderson, *Computational Fluid Dynamics. The Basics with Applications*, 1st ed., McGraw-Hill, New York, 1995.
[2] J. Blazek, *Computational Fluid Dynamics: Principles and Applications*, 2nd ed., Elsevier Science, San Diego, CA, 2005.
[3] T. Norton, D.-W. Sun, J. Grant, R. Fallon, and V. Dodd, *Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: A review*, Bioresour. Technol. 98 (2007), pp. 2386–2414.
[4] B.S. Massey, J. Ward-Smith, *Mechanics of Fluids*, 8th ed., Taylor & Francis, London, 2006.
[5] L. Yuan, A.C. Smith, and J.F. Brune, *Computational fluid dynamics study on ventilation flow paths in longwall gobs*, 11th U.S./North Am. Mine Vent. Symp. State College, PA, 2006, pp. 591–598.
[6] T.X. Ren and R. Balusu, *CFD modelling of goaf gas migration to improve the control of spontaneous combustion in longwalls*, Coal Oper. Conf., Wollongong, UNSW, Australia, 2005, pp. 259–264.
[7] H.K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics, The Finite Volume Method*, Prentice Hall, Malaysia, 1995.
[8] S.A. Silvester, *The integration of CFD and VR methods to assist auxiliary ventilation practice*, Fluid Dyn. The University of Nottingham, 2002.
[9] H.L. Hartman, J.M. Mutmansky, R.V. Ramani, and Y.J. Wang, *Mine Ventilation and Air Conditioning*, 3rd ed., Wiley-Interscience, New York, 1998.
[10] J. Moureh and D. Flick, *Airflow characteristics within a slot-ventilated enclosure*, Int. J. Heat Fluid Flow. 26 (2005), pp. 12–24.
[11] A. Bakker, The colorful fluid mixing gallery [Internet]. 2008. Available at http://www.bakker.org/cfm.
[12] L. Yuan and A.C. Smith, *Computational fluid dynamics modeling of spontaneous heating in longwall gob areas*, SME Annu. Meet. Denver, CO, Preprint 2007. pp. 07–101.
[13] L. Yuan, A.C. Smith, *Modeling the effect of barometric pressure changes on spontaneous heating in bleederless longwall panels*, SME Annu. Meet. Denver, CO, Preprint 2007, pp. 07–101.
[14] L. Yuan and A.C. Smith, *Numerical study on effects of coal properties on spontaneous heating in longwall gob areas*, Fuel. 87 (2008), pp. 3409–3419.
[15] L. Yuan and A.C. Smith, *Effects of ventilation and gob characteristics on spontaneous heating in longwall gob areas*, 12th US/North Am. Mine Vent. Symp. Reno, NV, 2008, pp. 141–147.
[16] G. Collecutt, D. Humphreys and D. Proud, *CFD simulation of underground coal dust explosions and active explosion barriers*, 7th Int. Conf. CFD Miner. Process Ind. Melbourne, Australia, 2009, pp. 1–6.
[17] J. Toraño, R. Rodriguez, I. Diego, J. Rivas, and A. Pelegry, *Influence of the pile shape on wind erosion CFD emission simulation*, Appl. Math. Model. 31 (2007), pp. 2487–2502.
[18] J. Toraño, S. Torno, M. Menéndez, and M. Gent, *Auxiliary ventilation in mining roadways driven with roadheaders: Validated CFD modelling of dust behaviour*, Tunnelling Underground Space Technol. 26 (2011), pp. 201–210.

[19] S. Torno, J. Toraño, M. Menéndez, and M. Gent, *CFD simulation of blasting dust for the design of physical barriers*, Environ. Earth Sci. 64 (2010), pp. 73–83.
[20] J. Aubin, D.F. Fletcher, and C. Xuereb, Modeling turbulent flow in stirred tanks with CFD: the influence of the modeling approach, turbulence model and numerical scheme, Exp. Therm. Fluid Sci. 28 (2004), pp. 431–445.
[21] P.T.L. Koh, M. Manickam, and M.P. Schwarz, *CFD simulation of bubble-particle collisions in mineral flotation cells*, Miner. Eng. 13 (2000), pp. 1455–1463.
[22] P.T.L. Koh and M.P. Schwarz, *CFD modelling of bubble–particle attachments in flotation cells*, Miner. Eng. 19 (2006), pp. 619–626.
[23] G.L. Lane, M.P. Schwarz, and G.M. Evans, *Predicting gas-liquid flow in a mechanically stirred tank*, Appl. Math. Model. 26 (2002), pp. 223–235.
[24] M.G. Vega, K.M.A. Argüelles Díaz, J.M.F. Fernández Oro, R.B. Tajadura, and C.S. Santolaria Morros, *Numerical 3D simulation of a longitudinal ventilation system: Memorial Tunnel case*, Tunnelling Underground Space Technol. 23 (2008), pp. 539–551.
[25] T. Skjold, R.K. Eckhoff, B.J. Arntzen, K. Lebecki, Z. Dyduch, R. Klemens, and P. Zydak. *Simplified modelling of explosion propagation by dust lifting in coal mines*, Proceeding 5th Int. Semin. Fire Exopl. Hazards. Edinburgh, UK, 2007, pp. 23–27.
[26] D.D. Ndenguma, *Computational fluid dynamics model for controlling dust and methane in underground coalmine*, Science (80-), University of Pretoria, 2010.
[27] S.A. Silvester, I.S. Lowndes, and S.W. Kingman, *The ventilation of an underground crushing plant*, Min. Technol. IMM Trans. Sect. A. 113 (2004), pp. 201–214.
[28] A. Mazzoldi, T. Hill, and J.J. Colls. *Assessing the risk for CO_2 transportation within CCS projects, CFD modelling*, Int. J. Greenh. Gas Control. Elsevier, 5 (2011), pp. 816–825.
[29] K.W. Moloney, I.S. Lowndes, M.R. Stokes, and G. Hargrave, *Studies on alternative methods of ventilation using Computational Fluid Dynamics (CFD), scale and full scale gallery tests*, Proc. 6th Int. Mine Vent. Congr., Pittsburgh, PA, USA, 1997, p. 7.
[30] S.B. Pope, *Turbulent Flows*, Cambridge University Press, New York, 2000.
[31] E.M. Marshall and A. Bakker, *Computational fluid mixing*, Fluid Dyn. Fluent Incorporated, 2001.
[32] V. Yakhot, S.A. Orszag, S. Thangam, T.B. Gatski, and C.G. Speziale, *Development of turbulence models for shear flows by a double expansion technique*, Phys. Fluids A Fluid Dyn. 4 (1992), pp. 1510–1520.
[33] K.G. Gebremedhin and B.X. Wu, *Characterization of flow field in a ventilated space and simulation of heat exchange between cows and their environment*, J. Therm. Biol. 28 (2003), pp. 301–319.
[34] P.A. Durbin. *A Reynolds stress model for near-wall turbulence*, J. Fluid Mech. 2006, 249, pp. 465–498.
[35] P. Spalart, S. Allmaras, *A one-equation turbulence model for aerodynamic flows*, Rech. Aerosp. 439(1) (1994), pp. 5–21.
[36] A.M. Wala, S. Vytla, C.D. Taylor, and G. Huang, *Mine face ventilation: A comparison of CFD results against benchmark experiments for the CFD code validation*, Min. Eng. 59 (2007), pp. 49–55.
[37] M. Parra, J. Villafruela, F. Castro, and C. Méndez, *Numerical and experimental analysis of different ventilation systems in deep mines*, Build. Environ. 41 (2006), pp. 87–93.
[38] A.P. Sasmito, E. Birgersson, H.C. Ly, and A.S. Mujumdar, *Some approaches to improve ventilation system in underground coal mines environment – A computational fluid dynamic study*, Tunnelling Underground Space Technol. 34 (2013), pp. 82–95.
[39] A.D. Gosman, *Developments in CFD for industrial and environmental applications in wind engineering*, J. Wind Eng. Ind. Aerodyn. 81 (1999), pp. 21–39.
[40] D.N. Sorensen and P.V. Nielsen, *Quality control of computational fluid dynamics in indoor environments*, Indoor Air. 13 (2003), pp. 2–17.
[41] J.D. Anderson, *Computational Fluid Dynamics. The Basics with Applications*, 1st ed., McGraw-Hill, New York, 1995.
[42] H. Lomax, T.H. Pulliam, and D.W. Zingg, *Fundamentals of Computational Fluid Dynamics*, Springer, New York, 2001.
[43] J.M. Berkoe and D.M. Lane, *Putting computational fluid dynamics to work on mining and metals projects*, SME Annu. Meet. Preprint, Salt Lake City, UT, USA, 2000, pp. 00–115.
[44] T.T. Bui, *CFD analysis of nozzle jet plume effects on sonic boom signature*, 47th AIAA Aerosp. Sci. Meet, Orlando, Florida, USA, 2009, pp. 1–28.
[45] G. Xu, E. Jong, K. Luxbacher, and S. Ragab, *Computational fluid dynamics study of tracer gas dispersion in a mine after different ventilation damage scenarios*, SME Annu. Meet. Seattle, WA, preprint 2012, pp. 12–051.
[46] I. Diego, S. Torno, J. Toraño, M. Menéndez, and G. Malcolm, *A practical use of CFD for ventilation of underground works*, Tunnelling Underground Space Technol. 26 (2011), pp. 189–200.
[47] I. Diego, S. Torno, and J. Toraño, *CFD simulation of aerodynamic resistance in underground spaces ventilation*, WIT Trans. Built Environ. (2008), 102, pp. 12–23.
[48] F.F. Peng and Y.K. Xia, *Analysis of a dense-medium separator for course coal separation using computational fluid dynamics*, Miner. Metall. Process. 24 (2007), pp. 1–12.

[49] F.F. Peng and Y. Xia, *Fluid dynamic modeling of fine particle separation in hindered-settling bed separators by CFD*, SME Annu. Meet. Denver, CO, preprint 2004, pp. 04–033.
[50] J. Toraño, S. Torno, M. Menendez, M. Gent, and J. Velasco, *Models of methane behaviour in auxiliary ventilation of underground coal mining*, Int. J. Coal Geol. Elsevier B.V., 80 (2009), pp. 35–43.
[51] A.M. Wala, J.C. Yingling, J. Zhang, and R. Ray, *Validation study of computational fluid dynamics as a tool for mine ventilation design*, Proc. 6th Int. Mine Vent. Congr., Pittsburgh, PA, 1997.
[52] J.B. Dick, *Measurement of ventilation using tracer gas*, Heating, Pip., USA Air Cond. 22 (1950), pp. 131–137.
[53] E.D. Thimons and F.N. Kissell, *Tracer Gas as an Aid in Mine Ventilation Analysis*, U.S. Bureau of Mines, Washington, DC, 1974.
[54] R.J. Timko and E.D. Thimons, *Sulfur Hexafluoride as a Mine Ventilation Research Tool – Recent Field Applications*, U.S. Bureau of Mines, Washington, DC, 1982.
[55] G. Xu, K.D. Luxbacher, S. Ragab, and S. Schafrik, *Development of a remote analysis method for underground ventilation systems using tracer gas and CFD in a simplified laboratory apparatus*, Tunnelling Underground Space Technol. 33 (2012), pp. 1–11.
[56] G. Xu, E.C. Jong, K.D. Luxbacher, S.A. Ragab, and M.E. Karmis, *Remote characterization of ventilation systems using tracer gas and CFD in an underground mine*, Saf. Sci. 74 (2015).
[57] I.M. Konduri, M.J. McPherson, and E. Topuz, *Experimental and numerical modeling of jet fans for auxiliary ventilation in mines*, Proc. 6th Int. Mine Vent. Symp., Pittsburgh, PA, USA 1997, pp. 505–510.
[58] R.B. Krog, S.J. Schatzel, H.N. Dougherty, *Airflow distribution patterns at a longwall mine depicted by CFD analysis and calibrated by a tracer gas field study*, SME Annu. Meet. Denver, CO, preprint 2011, pp. 11–067.
[59] F.H. Post and T. van Walsum, *Fluid flow visualization*, Focus Sci. Vis. 40 (1993), pp. 1–37.
[60] A. Wala, D. Turner, and J. Jacob, *Experimental study of mine face ventilation system for validation of numerical models*, Proceeding 9th North Am. U.S. Mine Vent. Symp, Kingston, Canada, 2002, pp. 191–196.
[61] D.M. Hargreaves and I.S. Lowndes, *The computational modeling of the ventilation flows within a rapid development drivage*, Tunnelling Underground Space Technol. 22 (2007), pp. 150–160.
[62] C.D. Taylor, R.J. Timko, E.D. Thimons, and T. Mal, *Using ultrasonic anemometers to evaluate factors affecting face ventilation effectiveness*, SME Annu. Meet. Salt Lake City, UT, preprint 2005, pp. 05–080.
[63] H. Hartmann, J.J. Derksen, C. Montavon, J. Pearson, I.S. Hamill, and H.E.A. van den Akker, *Assessment of large eddy and RANS stirred tank simulations by means of LDA*, Chem. Eng. Sci. 59 (2004), pp. 2419–2432.
[64] Z. Jaworski and J. Dudczak. *CFD modelling of turbulent macromixing in stirred tanks. Effect of the probe size and number on mixing indices*. Comput. Chem. Eng. 22 (1998), S293–S298.
[65] Z. Jaworski, K.N. Dyster, V.P. Mishra, A.W. Nienow, and M.L. Wyszynski, *A study of an up- and down-pumpling wide-blade hydrofoil impeller: Part II. CFD analysis*, Can. J. Chem. Eng. 76 (1998), pp. 866–876.
[66] J. Sheng, H. Meng, and R.O. Fox, *Validation of CFD simulations of a stirred tank using particle image velocity data*, Can. J. Chem. Eng. 76 (1998), pp. 611–625.
[67] A.M. Wala, J.C. Yingling, J. Zhang, and R. Ray, *Validation study of computational fluid dynamics as a tool for mine ventilation design*, Proceeding 6th Int. Mine Vent. Congr., Pittsburgh, PA, USA 1997.
[68] A.M. Wala, S. Vytla, G. Huang, and C.D. Taylor, *Study on the effects of scrubber operation on the face ventilation*. 12th U.S./North Am. Mine Vent. Symp, Reno, NV, USA 2008, pp. 281–286.
[69] A.M. Wala, J.C. Yingling, and J. Zhang, *Evaluation of the face ventilation systems for extended cuts with remotely operated mining machines using three-dimensional numerical simulations*. SME Annu. Meet. preprint, Orlando, Florida, USA, 1998, pp. 98–209.
[70] R.K. Jade and B.S. Sastry, *An experimental and numerical study of two-way splits and junctions in mine airways*. 12th U.S./North Am. Mine Vent. Symp, Reno, NV, USA 2008. pp. 293–298.
[71] Y. Zheng and J.C. Tien, *DPM dispersion study using CFD for underground metal/nonmetal mines*. 12th U.S./North Am. Mine Vent. Symp, Reno, NV, USA 2008, pp. 487–494.
[72] J.C. Kurnia, A.P. Sasmito, W.Y. Wong, and A.S. Mujumdar, *Prediction and innovative control strategies for oxygen and hazardous gases from diesel emission in undergroundmines*, Sci. Total Environ. 481 (2014), pp. 317–334.
[73] S.M. Aminossadati, *Numerical simulation of ventilation air flow in underground mine workings*. 12th U.S./North Am. Mine Vent. Symp, Reno, NV, 2008, pp. 253–260.
[74] X. Zhang, Y. Zhang, and J.C. Tien, *The efficiency study of the push-pull ventilation system in underground mine*, 2011 Undergr. Coal Oper. Conf., Wollongong, NSW, Australia, 2011.
[75] R.E. Ray, M.J. Gilbey, and P. Kumar, *The application of vertically-mounted jet fans in ventilation shafts for a rail overbuild*. 12th U.S./North Am. Mine Vent. Symp., Reno, NV, 2008, pp. 415–424.
[76] D.C. Panigrahi and D.P. Mishra, *CFD simulations for the selection of an appropriate blade profile for improving energy efficiency in axial flow mine ventilation fans*. J. Sustain. Min. 13(1) (2014), pp. 15-21.
[77] G. Xu, E. Jong, K. Luxbacher, and H.M. McNair, *Effective utilization of tracer gas in characterization of underground mine ventilation networks*, Process Saf. Environ. Prot. 99 (2015), pp. 1–10.
[78] G. Xu, J.R. Bowling, K.D. Luxbacher, and S.Ragab, *Computational fluid dynamics simulations and experimental validation of tracer gas distribution in an experimental underground mine*. 2011 SME Annu. Meet. Denver, CO, preprint 2011, pp. 11–121.

[79] E. Jong, S. Underwood, G. Xu, K. Luxbacher, and H. McNair, *A technique for creating perfluorocarbon tracer (PFT) calibration curves for tracer gas studies*, 14th U.S./North Am. Mine Vent. Symp. Salt Lake City, UT, 2012.
[80] S.W. Underwood, E.C. Jong, K.D. Luxbacher, E.A. Sarver, N.S. Ripepi, and H.M. McNair, *Solid phase microextraction (SPME) sampling under turbulent conditions and for the simultaneous collecting of tracer gases*, Int. J. Min. Sci. Technol. 25 (2015), pp. 559–563.
[81] E.C. Jong, K.D. Luxbacher, M.E. Karmis, and E.C. Westman. *Field test of a perfluoromethylcyclohexane (PMCH) permeation plug release vessel (PPRV) in an underground longwall mine*. Min. Technol. 2015, In Press.
[82] E.C. Jong, Y. Peng, W.T. Bradley, K.D. Luxbacher, Z. Agioutantis, and H.M. McNair, *Development of a perfluoromethylcyclohexane (PMCH) permeation plug release vessel (PPRV) for tracer gas studies in underground mines*, Process Saf. Environ. Prot. 95 (2015), pp. 136–145.
[83] E.C. Jong and K.D. Luxbacher, *An evaluation of a perfluoromethylcyclohexane (PMCH) permeation plug release vessel (PPRV) in a controlled turbulent environment*, Int. J. Min. Sci. Technol. 25 (2015), pp. 243–251.
[84] E.C. Jong, P.V. Macek, I.E. Perera, K.D. Luxbacher, and H.M. McNair, *An ultra-trace analysis technique for SF6 using gas chromatography with negative ion chemical ionization mass spectrometry*, J. Chromatogr. Sci. 6 (2015), pp. 1–6.
[85] R. Patterson and K.D. Luxbacher, *Tracer gas applications in mining and implications for improved ventilation characterisation*. Int. J. Mining, Reclamation, Environ. 26 (2012), 337–350.
[86] L. Yuan and A.C. Smith, *CFD modeling of spontaneous heating in a large-scale coal chamber*, J. Loss Prev. Process Ind. 22 (2009), pp. 426–433.
[87] A.C. Smith and L. Yuan, *Simulation of spontaneous heating in longwall gob area with a bleederless ventilation system*. SME Annu. Meet. Salt Lake City, UT, preprint 2008. pp. 08–043.
[88] J.C. Edwards and C.C. Hwang, *CFD modeling of fire spread along combustibles in a mine entry*, SME Annu. Meet. Littleton, CO, preprint 2006, pp. 06–027.
[89] J.C. Edwards, R.A. Franks, G.F. Friel, and L. Yuan, *Experimental and modeling investigation of the effect of ventilation on smoke rollback in a mine entry*, Soc. Min. Metall. Explor. 58 (2006), pp. 53–58.
[90] J. Huang, J. Bruining, and K.-H.A.A. Wolf, *Modeling of gas flow and temperature fields in underground coal fires*, Fire Saf. J. 36 (2001), pp. 477–489.
[91] G.B. Grant, S.F. Jagger, and C.J. Lea, *Fires in tunnels*, Philos. Trans. R. Soc. A Math. Phys. Eng. Sci. 356 (1998), pp. 2873–2906.
[92] M.A. Trevits, L. Yuan, K. Teacoach, M.P. Valoski, and J.E. Urosek, *Understanding mine fires by determining the characteristics of deep-seated fires*, NIOSH Document, Denver, CO, USA,2009.
[93] J.C. Edwards and C.C. Hwang, *CFD analysis of mine fire smoke spread and reverse flow conditions*. 8th U.S. Mine Vent. Symp. NIOSH Document, Rolla, MO, USA, 1999. pp. 417–422.
[94] C.C. Hwang and J.C. Edwards, *CFD modeling of smoke reversal*. Proc. Int. Conf. Eng. Fire Prot. Des. NIOSH Document, Bethesda, MD, 2001, pp. 376–387.
[95] S.F. Luchian and A.G. Bendelius, *West Virginia memorial tunnel fire ventilation test program*, Int. Conf. Fires Tunnels. Boston, MA, 1994.
[96] B.L. Warner, *Evaluation of potential fire hazard caused by exposed timber in mine passageways*, USBM Contract Final Rep. 1975.
[97] A. Kelsey, C.J. Lea, I.S. Lowndes, D. Whittles, and T.X. Ren, *CFD modelling of methane movement in mines*, 30th Int. Conf. Saf. Mines Res. Institutes, Johannesburg, Gauteng, South Africa, 2003, pp. 475–486.
[98] T.X. Ren, J.S. Edwards, and R.R. Jozefowicz, *CFD modelling of methane flow around longwall coal faces*. Proc. 6th Int. Mine Vent. Congr., Pittsburgh, PA, USA 1997, pp. 247–251.
[99] J.C. Kurnia, A.P. Sasmito, and A.S. Mujumdar, *Simulation of a novel intermittent ventilation system for underground mines*, Tunnelling Underground Space Technol. 42 (2014), pp. 206–215.
[100] K. Oraee and A. Goodarzi, *Mathematical modeling of coal seam methane drainage in longwall mining*, SME Annu. Meet. Preprint 2010, pp. 10–115.
[101] R. Balusu, N. Tuffs, D. White, and T. Harvey, *Surface goaf gas drainage strategies for highly gassy longwall*, J. Mine Vent. Soc. South Africa. 59 (2006), pp. 78–84.
[102] C.Ö. Karacan, T. Ren, and R. Balusu, *Advances in grid-based numerical modeling techniques for improving gas management in coal mines*. 12th US/North Am. Mine Vent. Symp., Reno, NV, USA 2008.
[103] G. Esterhuizen and C. Karacan, *A methodology for determining gob permeability distributions and its application to reservoir modeling of coal mine longwalls*, SME Annu. Meet. Denver, CO, 2007.
[104] K. Tanguturi and R. Balusu, *CFD modeling of methane gas distribution and control strategies in a gassy coal mine*, J. Comput. Multiph. Flows. 6 (2014), pp. 65–77.
[105] T.X. Ren, R. Balusu, and P. Humphries, Development of innovative goaf inertisation practices to improve coal mine safety, Coal Oper. Conf., Wollongong, NSW, Australia, 2005, pp. 315–322.
[106] R. Mossad, A. Vella, and R. Balusu, *Inertisation of highwall mining to control methane concentrations at the Moura Mine*, Seventh Int. Conf. CFD Miner. Process Ind. Melbourne, Australia, 2009, pp. 1–6.
[107] R. Balusu, P. Humpries, P. Harrington, M. Wendt, and S. Xue, *Optimum inertisation strategies*, Queensl. Min. Ind. Heal. Saf. Conf. Queensland, Australia, 2002, pp. 133–144.

[108] M. Ramakrishna, B. Rao, T. Krishna, and T. Ren, *Inertisation options for BG method and optimisation using CFD modelling*, Int. J. Min. Sci. Technol. 25 (2015), pp. 401–405.
[109] A. Vella, *Ventilation of highwall mining to control methane concentration at the Moura Mine*, Mech. Eng. The University of Southern Queensland, 2006.
[110] M.A. Trevits, L. Yuan, M. Thibou, and G. Hatch, *Use of CFD modeling to study inert gas injection into a sealed mine area*, SME Annu. Meet. Preprint, Phoenix, AZ, USA 2010, pp. 10–207.
[111] J. Van Heerden and P. Sullivan, *The application of CFD for evaluation of dust suppression and auxiliary ventilating systems used with continuous miners*. Proc. 6th US Mine Vent. Symp. SME, Littleton, 1993, pp. 293–297.
[112] R.B. Srinivase, E.Y. Baafi, N.I. Aziz, and R.N. Singh, *Three dimensional numerical modelling of air velocities and dust control techniques in a longwall face*. Proceeding 6th U.S. Mine Vent. Symp. Salt Lake City, UT, 1993.
[113] D.M. Hargreaves and I.S. Lowndes, *The evaluation of the design and operation of alternative auxiliary ventilation layouts for use with continuous miner systems in rapid development drivages*. Final Rep. 7220-AC-857, Comm. Eur. Communities, Dir. Energy Transp, 2001.
[114] S.A. Silvester, I.S Lowndes, and D.M.Hargreaves, *A computational study of particulate emissions from an open pit quarry under neutral atmospheric conditions*, Atmos. Environ. Elsevier, 43 (2009), pp. 6415–6424.
[115] T. Ren and R. Balusu, *The use of CDF modelling as a tool for solving mining health and safety problems*, Coal Oper. Conf., Wollongong, NSW, Australia, 2010, pp. 339–349.
[116] D. Proud, G. Collecutt, and D. Humphreys, *Computational fluid dynamics modelling of coal dust explosions and suppression systems*, Third Aust. Mine Vent. Conf. The Australasian Institute of Mining and Metallurgy, Sydney, Australia, 2015, pp. 309–313.
[117] G. Collecutt, D. Humphreys, and D. Proud, *CFD simulation of underground coal dust explosions and active explosion barriers*, Seventh Int. Conf. CFD Miner. Process Ind. CSIRO Australia, Melbourne, Australia, 2009, pp. 1–6.
[118] J. Lichter, A.V. Potapov, and R. Peaker, *The use of computational fluid dynamics and discrete element modeling to understand the effect of cell size and inflow rate on flotation bank retention time distribution and mechanism performance*, Proc. 39th AGM Can. Min. Proc., Ottawa, Ontario, Canada, 2007, pp. 473–496.
[119] F.D. Ardejani, B.J. Shokri, E. Baafi, K.S. Panahi, and R.N. Singh, *Application of Computational Fluid Dynamics (CFD) for simulation of acid mine drainage generation and subsequent pollutants transportation through groundwater flow systems and rivers*. INTECH Open Access Publisher, 2011.
[120] A. Mazzoldi, T. Hill, and J. Colls, *A consideration of the jet-mixing effect when modelling CO_2 emissions from high pressure CO_2 transportation facilities*. Energy Procedia Elsevier, 1 (2009), pp. 1571–1578.
[121] C. Dixon, O. Heynes, and M. Hasson, *Assessing the Hazards Associated with Release and Dispersion of Liquid Carbon Dioxide on Offshore Platforms*, MMI Engineering, Warrington, 2008.