ELSEVIER

Available online at www.sciencedirect.com

SciVerse ScienceDirect

Procedia Engineering 37 (2012) 125 – 131

Procedia Engineering

www.elsevier.com/locate/procedia

The influence of mesh generation on the results of flow solving in working face and gob

Jianliang Gao [a,b,] *, Dandan Zhang [a,b], Ming Yang [a,b]

and Xuebo Zhang [a,b]

[a] *School of Safety Science & Engineering, Henan Polytechnic University,Jiaozuo , Henan, 454003 ,China*

[b] *State Key Laboratory Cultivation Base for Gas Geology and Gas Control ,Jiaozuo , Henan, 454003 ,China*

Abstract

The distribution of air leakage between working face and gob area and migration of gob gas are simulated by using commercial CFD software Fluent. Mesh generation is one of the most important factors influencing the calculation accuracy of airflow field in gob area. If a coarse grid is used to simulate, the accurate results cannot be obtained. However, using a finer grid need to spend much more time. In this paper, near-wall boundary layer mesh and various grid spacing of working face and gob are constructed with Gambit and respectively with Fluent simulation near-wall boundary layer encryption and the airflow field and migration of gob gas with different grid spacing in working face and gob. Compared with the original models: the eddy zone range of the upper corner and down corner and the highest gas accumulation concentration of the upper corner can be simulated accurately, which use boundary layer encryption and reasonable grid spacing .This results have important significance to deal with gas accumulation in the upper corner of working face and choose the reasonable gas drainage in gob.

Keywords: Gob; Mesh generation;Numerical simulation; Airflow field;Gas distribution

1. Introduction

With the rapid development of mining production technology, mine production efficiency has been improving greatly in recent years. However, due to the enhancement of mining intensity of fully mechanized coal face, amount of gas emitted into working from coal seam and gob increases. Therefore, it is important to master the gas distribution pattern in gob in order to take the reasonable measures to prevent the gas accumulation.

A number of studying on airflow field in gob area, migration of gob gas and related safety problem has been presented. Michael & Vlasseva assumed that the gob airflow was turbulent, made numerical simulations to study pressure, velocity distribution and percolation in the waste [1]. A.D.Jones, Lowrie.S.J.R etc studied the distribution of air leakage between working face and gob area and migration of gob gas by using aerodynamics proportion model and model experiment, and obtained the distribution of gob gas [2]. Jiachen Wang [3] & Qianting Hu [4] studied the distribution of the flow field, methane concentration at coal face under U type ventilation by means of numerical simulation software under

* Corresponding author. Tel.: 0319-3987882; fax: 0319-3987882
E-mail address: gao@hpu.edu.cn

doi:10.1016/j.proeng.2012.04.214

assumption that the gob porous media was isotropic based on Darcy's rules. Jianliang Gao & Haisheng Wang [5] & Jiajia Liu [6] simulated the distribution of air leakage between working face and gob area, and analyzed the quantity and speed of air leakage, the position and width of the spontaneous combustion "Three Zone" under condition of different distribution of permeability in gob. Numerical simulation has been applied widely to study the airflow in working face, gas migration and emission in coal seam, distribution of air leakage and gas in gob. It is well known that the quality of grids has great effects the results of the numerical simulation. So the technique of grid generation becomes a key to the results with reasonable accuracy. If the coarse grid is used, accurate results cannot be obtained. However, using the finer grid needs to spend much more time. As for complicated model, mesh generation methods in different zone has important influence on the quality of mesh.

In this paper, the mesh of working face and gob was established by applying the methods of grid separate blocked generation with Gambit. Boundary layers mesh was applied to the near-wall of working face, then the working face and gob with different grid spacing were simulated for the distribution of air leakage and the migration of gob gas. The reasonable grid spacing was determined to get the more reliable flow field of working face and gob.

2. Mathematical model of gob gas migration

2.1. Assumptions

The gob porous media is approximately regarded as isotropic. 2) Gob gas is considered as incompressible fluid gas. The heat dissipation power caused by fluid viscous can be ignored and its flow is approximately a stable and isothermal process. 3) The gob gas is a mixture of methane and air. The gas diffusion coefficient is constant. 4) Gob gas flow in porous media follows linear infiltration rules, Darcy's rules [7].

2.2. Mathematical model

Airflow in gob obeys the continuity equation and momentum equations. Based on the above assumptions, the control equations of the gob gas migration mathematical model can be set:

The continuity equation:

$$\frac{\partial \rho u}{\partial x}+\frac{\partial \rho v}{\partial y}=S_m \tag{1}$$

Momentum (Navier-Stokes) equations:

$$\frac{\partial(n\rho uu)}{\partial x}+\frac{\partial(n\rho uv)}{\partial y}=\frac{\partial}{\partial x}(n\mu_{eff}\frac{\partial u}{\partial x})+\frac{\partial}{\partial y}(n\mu_{eff}\frac{\partial u}{\partial y})-n\frac{\partial p}{\partial x}-\frac{\mu}{e}u \tag{2}$$

$$\frac{\partial(n\rho vu)}{\partial x}+\frac{\partial(n\rho vv)}{\partial y}=\frac{\partial}{\partial x}(n\mu_{eff}\frac{\partial v}{\partial x})+\frac{\partial}{\partial y}(n\mu_{eff}\frac{\partial v}{\partial y})-n\frac{\partial p}{\partial y}-\frac{\mu}{e}v \tag{3}$$

Components transfer equation:

$$\frac{\partial(\rho c_s u)}{\partial x}+\frac{\partial(\rho c_s v)}{\partial y}=\frac{\partial}{\partial x}(D_s\frac{\partial \rho c_s}{\partial x})+\frac{\partial}{\partial y}(D_s\frac{\partial \upsilon c_s}{\partial y})+S_s \tag{4}$$

RNG $k-\varepsilon$ equations:

$$\frac{\partial(\rho k u)}{\partial x}+\frac{\partial(\rho k v)}{\partial y}=\frac{\partial}{\partial x}(\alpha_k\mu_{eff}\frac{\partial k}{\partial x})+\frac{\partial}{\partial y}(\alpha_k\mu_{eff}\frac{\partial k}{\partial y})+G_k-\rho\varepsilon \tag{5}$$

$$\frac{\partial(\rho\varepsilon u)}{\partial x}+\frac{\partial(\rho\varepsilon v)}{\partial y}=\frac{\partial}{\partial x}(\alpha_\varepsilon\mu_{eff}\frac{\partial \varepsilon}{\partial x})+\frac{\partial}{\partial y}(\alpha_\varepsilon\mu_{eff}\frac{\partial \varepsilon}{\partial y})+\frac{\varepsilon}{k}\left(C_{1\varepsilon}^{*}G_k-C_{2\varepsilon}\rho\varepsilon\right) \tag{6}$$

In the formula:

$$\mu_t = \rho C_\mu k^2 / \varepsilon \tag{7}$$

Where $C_{1\varepsilon} = 1.44$ $C_{2\varepsilon} = 1.92$ $C_\mu = 0.09$ $\sigma_k = 1.0$ $\sigma_\varepsilon = 1.3$

u, v—respectively for speed of x, y direction; ρ—gob gas flow density; μ—dynamic viscosity coefficient; n—porosity of porous medium; S_m—mass source term; C_s—the volume concentration of components; S_s—components productivity; ρc_s—the mass concentration; D_s—the components of diffusion coefficient; k is turbulent kinetic energy, m2/s2; ε is turbulent kinetic energy dissipation rate, m2/s3; G_k are generation items of turbulent kinetic energy k caused by the average velocity gradient. Physical model in gob gas migration.

2.3. Physical model in gob gas migration

The physical model can be established with Gambit software as shown in Fig.1—based on actual situation of working face and gob. The inlet and outlet roadways section of working face are set for 2.5m in width and 20m in length, the working face is 5m wide and 100m long, gob area is 100m long.

The entrance is set to be inlet boundary conditions (Velocity-inlet) with velocity of 2.5m/s, and exit is set to be outlet boundary conditions for freedom flow (Out-flow). Working face and gob area boundary are set to be internal boundary (Interior). Methane gas is released from the whole gob. The porosity porous media in gob is 0.3 with uniform distribution.

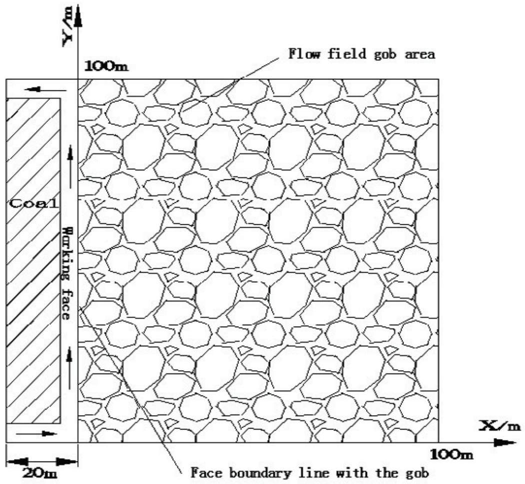


Fig. 1 The physical model of working face and gob

2.4. Meshing

Finite-volume method is used in simulation. The degree of attractive feature in fluid flow of working face and gob is determined by the density and distribution of nodes. The mesh of working face and gob can be established by applying the method of grid separate blocked generation with Gambit. Boundary layers mesh is applied to the near-wall surface of the roadway. Boundary layers are used to grow layers of cells of desired height from specified boundaries of geometry and are typically used to capture near-wall phenomena such as turbulence. Boundary layers mesh can be created by inputting any three of following parameter values: first row, growth factor, rows, depth. Boundary layers mesh in this case is generated as follows: height of first row of elements was 0.05m, factor for geometric series was 1.2, total number of element rows was 5, total height of boundary layer was 0.372m. The grid spacing in working face and gob are 0.1m, 0.5m; 0.2m, 0.5m; 0.2m, 1m; 0.2m, 2m, separately.

3. Simulation result analysis

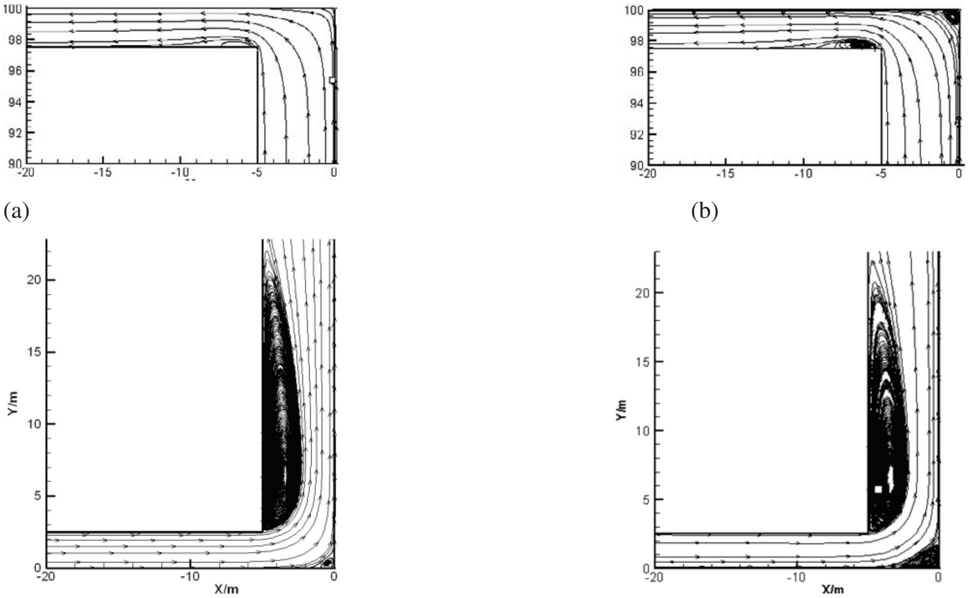
3.1 The effect of working face mesh generation on the numerical simulation of fluid

When the flow fields is turbulent under large reynolds number, viscous force can be ignored in farther from the wall because it is much lower than inertial force, however, it can not be ignored in the thin layer of near-wall where there is a quite large velocity gradient along the normal orientation of wall surface .This thin layer is called boundary layer. To simulate the flow fields of near-wall, the total height of boundary layer can be calculated and then create boundary layers mesh. The calculation formula of the total height of boundary layer is as follow:

$$\delta = 5\sqrt{\frac{vl}{v_{in}}} \tag{8}$$

Where: δ—the total height of boundary layer; *v*—dynamic viscosity coefficient; *l*—the length of the working face; v_{in} — inlet velocity.

The distributions of airflow on the upper corner of working face are shown in Fig.2 (a) and Fig.2 (b). The distributions of airflow on the bottom corner of working face are shown in Fig.2 (c) and Fig.2 (d). In Fig.2 (a) and Fig.2 (c) near-wall boundary layer mesh is used, and in Fig.2 (b) and Fig.2 (d) boundary layers mesh is not applied.

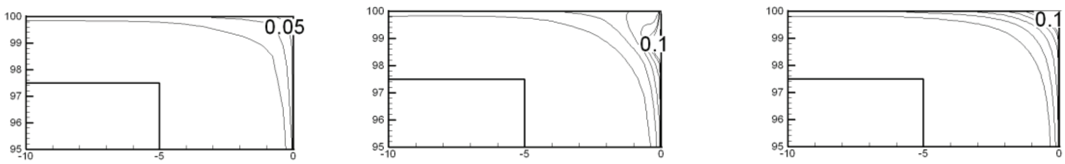


(a) (b) (c) (d)

Fig. 2 (a) Airflow field on upper corner not using near-wall boundary layer mesh; (b) Airflow field on upper corner using nearwall boundary layer mesh; (c) Airflow field on bottom corner not using near-wall boundary layer mesh; (d) Airflow field on bottom corner using near-wall boundary layer mesh

From the Fig.2, it can be seen that there are obvious eddy zones in the upper corner and bottom corner by using boundary layer mesh. However, if boundary layer mesh is not used, the eddy zones can not be simulated reasonably. Bondary layer mesh has been proved to be an effective way to solve near-wall flow of working face and flow state on the upper corner and down corner zones where velocity changes greatly.

Using boundary layer mesh can solve the flow state which change greatly in the near-wall, but there is another question that how to determine the gird spacing to meet the solution requirement for obtaining the reliable results. In this part, the suitable gird spacing of working face can be determined by comparing the result of the numerical simulation of fluid where its gird spacing L is 0.4m, 0.2m, 0.1m, respectively. The solution results are shown in Fig.3.



(a) (b) (c)

Fig. 3 (a) Distribution of gas concentration (L=0.4m); (b) Distribution of gas concentration; (L=0.2m) (c) Distribution of gas concentration (L=0.1m)

In theory, the more finite the grid is, the more precise the solution results are. Fig.3 shows that when the gird spacing in zone of working face is 0.4m, the largest gas concentration in the upper corner is 5%. When the gird spacing in zone of working face is 0.2m, the largest gas concentration in the upper corner is 10%. When the gird spacing of working face is 0.1m, the largest gas concentration in the upper is almost same as the result of 0.2m. Therefore, gird spacing in zone of working face 0.2m can meet the solution requirement for obtaining the reliable results.

3.2 The effect of gob mesh generation on the numerical simulation of fluid

Under the condition of same boundary conditions and the grid spacing in zone of working face which is 0.2m, the simulation results with different spacing in zone of gob is investigated. The simulation results of gas concentration distribution when spacing in zone of gob are 2m, 1m, 0.5m, 0.25m are shown in Fig.4. The figures show that the gas distributions in gob a few meters away of the boundary between working face and golf have little difference. Near the boundary between working face and gob, especially close to the return airway the profile lines of gas distribution seems are dense when the spacing is 2m and 1m (Fig.4 (a) and (b), which cannot demonstrate the actual gas distribution near the boundary of working face and gob. The grids near the boundary should be finer. Fig.4 (c) and Fig.4 (d) show that both the spacing of 0.5m and 0.25m can get the well simulation results near the boundary of working face and gob. Therefore the spacing of 0.5m can obtain more reasonable accurate simulation results.

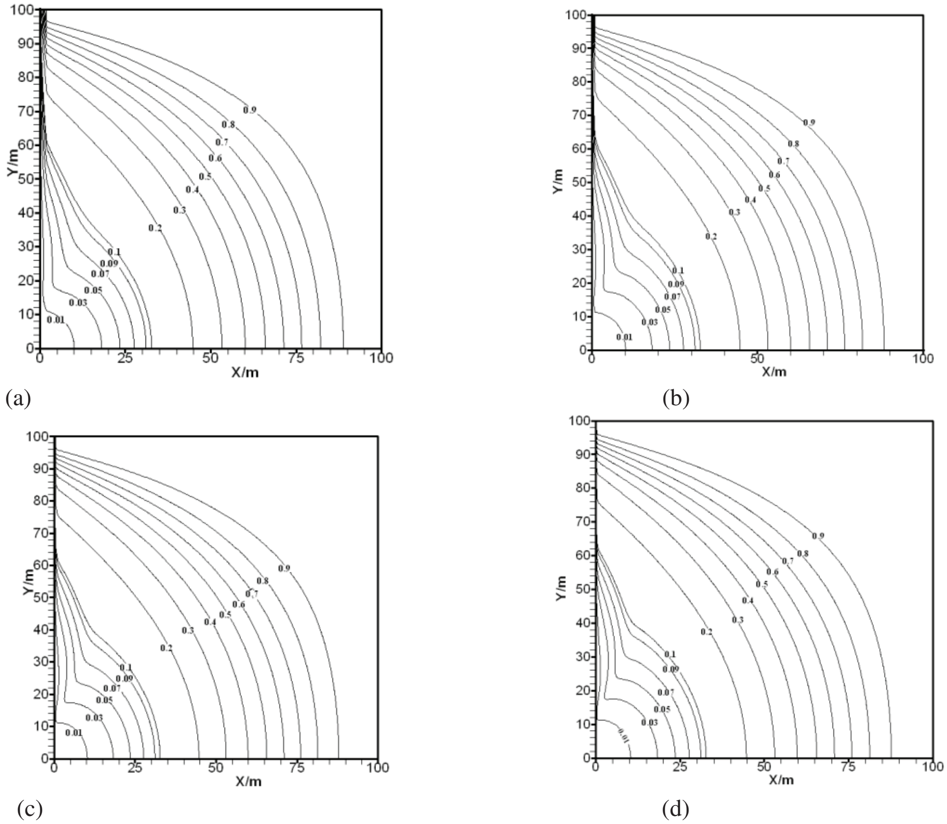


Fig. 4 (a) Distribution of gas concentration in gob (M=2m); (b) Distribution of gas concentration in gob (M=1m); (c) Distribution of Gas concentration in gob (M=0.5m); (d) Distribution of Gas concentration in gob (M=0.25m)

Conclusion

The airflow field and gas distribution are simulated in working face and gob with different grid spacing. The reasonable grid spacing w0as determined to get the more reliable flow field of working face and gob. Applying boundary layers mesh to the near-wall of working face, can obtain more reliable flow field of working face. The eddy zone range of working face and the gas accumulation on the upper corner can be simulated accurately by using boundary layer meshing techniques in working face. Making the gird spacing of 0.2m in zone of working face and spacing of 0.5m in zone of gob can obtain the numerical simulation results of airflow field and gas distribution in working face and golf with favourite accuracy.

Acknowledgements

The research project is sponsored by National Nature Science Foundation of China (Grant No. 51174079) and Research Fund for the Doctoral Program of Higher Education of China (Grant No. 20104116110001). Thanks to NSFC and RFDP for the financial support.

References

[1] Michael, Vlasseva. On Void Aerodynamics In a Porous Media Of a Gob Area. Proceedings Of the 7th US Mine Ventilation Symposium; 1995.
[2] Jones A D, Lowrie S J R. A Physical Scale Model of Flows In the Waste Of a Retreat Long wall Coalface. Proceedings Of the 6th International Mine Ventilation Congress; 1997.
[3] Jiachen Wang, Hongbao Zhao. Numerical Simulation Study On Flow Field Characteristics of Mining Face Spaces. Metal Mine; 2010, pp. 151-154.
[4] Qianting Hu, Yunpei Liang, Jianzhong Liu. CFD Simulation Of Gob Gas Flow Patterns. Journal of China Coal Society; 2007, pp. 719-723.
[5] Jianliang Gao, Haisheng Wang. Influence Of Permeability Distribution On Airflow Field of Leakage In Gob. China Safety Science Journal; 2010, pp. 81-85.
[6] Jianliang Gao, Jiajia Liu. Research On Flow Field and Gas Distribution in Gob Of Full-mechanized Mining Working Face; 2011.
[7] Jacob Bear. Dynamics Of Fluids In Porous Media. American Elsevier Publishing Company. INC; 1972.