Available online www.jsaer.com

Journal of Scientific and Engineering Research, 2018, 5(10):37-42

ISSN: 2394-2630
CODEN(USA): JSERBR

Research Article

Turbulence Numerical Simulation Method and Its Application in Mine Flow Field

ZHANG Zhi-gang, ZHU Xu-dong*

School of Safety Science and Engineering Henan Polytechnic University Jiaozuo Henan 454003, China

Abstract In order to study the advantages and disadvantages of different turbulence models and the applicable conditions, through the development of numerical simulation of turbulence, three methods of direct numerical simulation(DNS), Reynolds averaged Navier-Stokes (RNAS) and large eddy simulation (LES) are comprehensively reviewed. The actual situation in the flow field in the coal mine is combined with the research of the students. The application of the k-ε two-equation model in the Reynolds average numerical simulation in the mine flow field is analyzed from three angles of different ventilation methods, different ventilation systems and other related ventilation flow fields. Finally, the two equation model is perfected, the application field of the mine is expanded and the simulation calculation is improved. The corresponding suggestions and countermeasures are put forward.

Keywords Turbulence; Numerical simulation; Turbulence model; k-ε two-equation model; Mine flow field

1. Introduction

Turbulent flow is a complex three-dimensional flow state of fluid. Due to its motion law and the disorder and kinetic energy and diffusion of moving elements, turbulence has been studied a lot, but it has not yet formed a set. The mature theory is mainly to use semi-empirical formulas. The systematic study of turbulence by modern theories and methods began in the late 19th century, and O. Reynolds proposed that the statistical averaging method is the starting point for turbulence research [1]. For more than a century, although turbulence research has made great progress in the understanding of turbulent nature and practical applications, with the continuous improvement of computational fluid dynamics and computational aerodynamics, computer performance continues to improve, turbulence The numerical simulation method has become one of the bottlenecks that hinder people from applying the NS equation [2]. Up to now, the numerical simulation methods of turbulent motion can be roughly divided into: direct simulation (DNS), Reynolds average numerical simulation (RAMS), large eddy simulation (LES), research on the advantages and disadvantages of different turbulence models and their respective applications. Conditions are of great benefit to promoting the improvement of turbulence models and improving their application in different fields.

Numerical Simulation Method of Turbulence

Direct Simulation

The direct simulation method is a direct calculation of the NS equation in the calculation area. It is not necessary to establish any turbulence model. It is necessary to simultaneously solve the turbulence of the fluid in space and time. The overall scale of the turbulence is on the one hand and the fluid kinematic viscosity v and velocity kinetic energy. The rate of dissipation is related to, on the one hand, the boundary conditions on the calculation area. Direct numerical simulation is an effective means to study the theory of turbulence. The "numerical test" of fluids can obtain information that is difficult or impossible to obtain from the laboratory. In addition, direct simulation can be further developed into the actuality of turbulence models. Application, simulation results can be used to test existing turbulence models, explain the nature of turbulence, and deepen people's comprehensive understanding of turbulence. However, for turbulent flow with large Reynolds number, the disturbance of the fluid is more severe and irregular, and the resolution of space and time is higher. The calculation amount is very large, which requires the computer to have superior calculation ability [3]. The current computer capability can only calculate large turbulent vortices, and can not capture small eddies. Therefore, the direct numerical simulation method can only calculate the turbulent motion with less Reynolds number. Even if more accurate calculations can be made in the future, in order to make the simulation results more meaningful, a large number of calculation results need to be statistically processed.

Direct simulation is a method of accurately numerically simulating turbulence, so that all information about the turbulent flow field can be obtained, and experimental measurements cannot be fully realized. And because the N-S equation is directly simulated, there is no closure problem, and in principle all turbulence problems can be solved. However, under the limitations of existing computer capabilities, only low and low Reynolds number Re and simple geometric boundary turbulent motion can be simulated. The application field is mainly exploratory basic research of turbulence.

Reynolds Average Numerical Simulation

The Reynolds average numerical simulation method firstly performs time-averaging processing on the unsteady NS equation, and obtains a set of time-average equations. The unknowns in the equation are more than the number of equations, even if the time-average equation is further time-averaged, only corresponding. The higher order unknowns, so that the time-average equation can not be closed, and the correct solution is not obtained. Therefore, in engineering, it is necessary to make some assumptions about the time-average equation to make it closed for solving. The Reynolds time-average equation method is an effective method [4]. The Reynolds time-average equation divides the turbulent flow into time-average motion and pulsation motion. All the parameters in the turbulent flow are regarded as the combination of time average and pulsation value. The currently widely used turbulence models are: zero-equation turbulence. Model, an equation turbulence model, a two-equation turbulence model and a Reynolds stress equation model.

The zero-equation model does not need to add any other equations. Only by substituting the Reynolds stress into the equation based on the continuity equation and the momentum equation, some physical parameters can be expressed. However, the parameters of the zero-equation model are limited by the flow form of the fluid, the calculation result is related to the grid form, and has not been widely used in engineering. Different from the zero-equation model, an equation model overcomes the defect that the zero-equation model does not take into account turbulent convection and diffusion. It adds a kinetic energy k to the continuity equation and the momentum equation. Equations, such models can be used to separate flows. The simulation results are not affected by the mesh. They can also be used for predictive conversion, but they cannot handle the problems of planar jets and circular jets, and the accuracy is difficult to meet the requirements. The two-equation model is proposed on the basis of an equation model. It adds a governing equation about the dissipation rate ε to close the equations. This model deals with isotropic turbulence due to its simple form and applicability. There are obvious advantages in the flow problem, but there are also defects in the empirical constant uncertainty and the inapplicability to various flows such as low Reynolds number flow [5]. The Reynolds stress equation model, also known as the second moment closed mode, establishes the Reynolds stress transport equation, derives the turbulent stress formula from the pulsating velocity field, and then rationally simplifies the equation to close the equations. Complex three-dimensional anisotropic turbulent flow can be described, but its simulated nature of free shear flow is the same as that of the two-equation model. It is also difficult to predict the axisymmetric jet and wake as the Reynolds stress mode.

Large Eddy Simulation

The turbulent large eddy simulation method is a turbulent numerical simulation method between the turbulent direct numerical simulation method and the Reynolds average numerical simulation method, which becomes an effective method for turbulent mesoscopic numerical simulation with accuracy and acceptable calculation amount [6]. The basic idea of large eddy simulation is to accurately solve all turbulent motions in the entire computational region, thus including non-steady processes such as non-steady state, large-scale effects, and quasi-order structures that are not considered by other numerical simulation methods. The large eddy simulation calculates the large-scale eddy and the small-scale vortex separately. Using the small vortex almost isotropic and many commonalities, they can be treated with a uniform similarity model, and the large eddy is directly calculated. This method is highly efficient, but it still requires huge computation time and huge computer memory consumption. Its application in actual simulation needs further development.

2. Application of two-equation model in mine flow field

According to the characteristics and applicable conditions of different models, combined with the characteristics of coal mine flow field, when numerical simulation of downhole gas and dust is carried out, most of the two equations are selected. However, many scholars in the mine flow field are mostly focused on different Simulation of ventilation methods, different ventilation systems and other related ventilation flow fields.

Different Ventilation Methods

Tang Minbo's thesis: Numerical simulation study of the inflow ventilation flow field in the tunneling face [7]. According to the actual situation of the wind flow and the flow field distribution of the roadway, the flow field of the tunnel is built by the Fluent software. The physical model and mathematical model are analyzed and the boundary conditions are calculated. At the same time, the RNG k-ε model and the wall function method are used to describe the flow process of the roadway, and the numerical simulation of the tunneling process is carried out. The flow field of the tunneling in the localized ventilation is obtained. Distribution law and law of variation of wind flow rate. On the basis of the credible numerical simulation, the variation law of the airflow flow field and the central axis velocity of the air duct outlet under the condition of different limited attached jet characteristics of the roadway is studied. Yao Haifei et al.: Numerical simulation of the drafting ventilation flow and dust migration law [8]. Using the two-equation model, the velocity and pressure distribution of the X, Y and Z directions in the roadway were studied in the Fluent software. The dust migration trajectories of different particle sizes have obtained: less dust diffusion during extraction ventilation, and dust generated by excavation is all sucked into the negative pressure air cylinder, and the exhaust ventilation is more beneficial to control the dust in the roadway, especially the driver's seat. The content protects the health of workers such as drivers and other workers around the roadheader. Professor Zheng Jianguo and other papers: The study on the distribution of dust under the long-short-short ventilation system in the rock roadway[9], ignoring the energy dissipation caused by the viscous force of the wind flow, and considering the wind flow in the wall of the comprehensive excavation as incompressible heat insulation, The isothermal fluid is controlled by the three-dimensional steady-state incompressible NS equation and the standard k-ε double-equation turbulent flow model. At the same time, considering the gravity effect of the solid particles, the distribution of the wind field and dust concentration field in the fully mechanized excavation face is carried out. Numerical simulation study, the distribution law of wind flow field, the three-dimensional distribution law of dust concentration and the three-dimensional distribution law of dust concentration, and compare the simulation results with the measured results, which provides a reference for the treatment of dust. Wang Weijian's thesis: Numerical simulation and experimental study of pumping and post-mixing mixed ventilation in front of excavation work [10], using standard k-ε double equation model to solve, from ventilation flow field structure, dust migration law and dust control effect In the face of the excavation work, the pumping and mixing ventilation method was used to carry out the system research, and the reasonable ventilation parameters of the mixed ventilation were obtained. From the perspective of safety and economic benefits, the optimal air suction ratio was obtained.

Different ventilation systems

Feng Pujin's thesis: Study on the flow field and gas distribution law of the goaf in the Y-type ventilation working face [11], using the standard k-ε model to solve the control equations of the air leakage and gas distribution in the goaf, using the Fluent software to double the The value of the air leakage, the air leakage, the air leakage and the gas concentration distribution in the goaf are different under the different ratios of the return air volume and the return air volume in the airway. Simulation study. Yan Liangfeng's thesis: Numerical simulation of gas flow field in goaf under Y-type ventilation [12], based on the two-equation model, using flow field simulation software Fluent for gob of fully mechanized caving face under two-in-one Y-type ventilation The gas flow field and the concentration field distribution were numerically simulated, and the effects of different air volumes on the flow field were simulated. The numerical simulation results are comprehensively analyzed and studied. The gas flow and concentration distribution in the goaf of the fully mechanized caving face under two-in-one Y-type ventilation mode are obtained, which provides a reliable theoretical basis for gas control in the goaf. Professor Wang Kai and other papers: Numerical simulation of flow field and gas migration in J-type ventilated fully mechanized caving mining area [13], using the k-ε model to establish a J-type ventilation goaf for the new J-type ventilation system of fully mechanized caving mining Computational fluid dynamics model of zone flow field simulation. Through numerical simulation, the flow field and gas migration characteristics of the goaf under J-ventilation and U-ventilation conditions are compared and analyzed. The flow field and gas migration law of J-type ventilation goaf are systematically studied under the conditions of different goaf size, drainage capacity of gas-discharging lanes and installation position of extraction fan.

Simulation of other relevant ventilation flow fields

Professor Gao Jianliang and other papers: the influence of the position of the first layer of grid nodes on the calculation results of the local ventilation flow field [14], using the RNG k-ε double equation turbulence model, the first layer of grid nodes are arranged at different positions. Simulation calculation analysis shows that when using the wall function method to simulate the wind flow field of the local ventilation tunneling face, the first layer node at the near wall surface should be in the logarithmic layer, and the proportion in the logarithmic layer is more. The more accurate the results of the settlement, the better the simulation will be. Professor Wang Haiqiao and other papers: Numerical simulation of the flow field of the jet flow in the tunnel with a single headway [15], the turbulent k-ε double equation model of the indented restricted wall jet ventilation of the single headway is established, and the computational fluid dynamics is calculated. The software simulates the three-dimensional flow field of jet flow and ventilation in the single headway, and obtains the flow field law of the typical section of the bounded jet ventilation with limited space in the single headway. It is shown that the pressure-inflow flow field of the single-head road can be divided into the auxiliary jet flow zone, the impact jet attachment zone, the recirculation zone and the vortex zone. The numerical simulation is consistent with the experimental results, in order to study the monolithic roadway mass transfer process and gas transportation. The law of movement and efficiency of ventilation and drainage provide a theoretical basis. Professor Liu Ronghua and other papers: Numerical study on the characteristics of the flow field of the rotating jet shielding ventilation in the mining face [16], using the k-ε double equation turbulence model, the turbulent flow under the shielded ventilation mode and the wind flow field and harmful Numerical simulation of the material control situation shows that the rotating jet shielding ventilation can form a rotating air curtain in front of the excavation work, and the dust generated by the excavation is blocked in the limited space between the rotating air curtain and the tunneling end, and The suction port is discharged under the action of the suction airflow; under the combined action of the cyclone flow field and the suction airflow of the suction cylinder, the pressure at the suction port of the center of the roadway in front of the tuyere of the dust collection and dust removal device is lower than the surrounding pressure, so that the airflow is directed to the suction cylinder The collection greatly improves the suction effect of the suction port.

3. Suggestions and Countermeasures

Improve the two-equation model and explore new simulation methods

The k-ε double-equation turbulence model can achieve satisfactory results for non-separating flows, such as free shear flow and wall shear flow. For complex flows, such as flow separation or irregular boundaries, k-ε double equation turbulence Poor model [17]. In order to improve the accuracy of the model, it is necessary to explore the improved method of the equations used in the future, simplify the calculation steps, and improve the calculation of the model based on the original low Reynolds number model, wall function method and regional model in the processing method of the equation model. The solution conditions and calculation methods are used as an important basis for improving the model. In addition, some new angles can be explored for numerical simulation of turbulence, such as probability density function method, renormalization group theory, and methods based on Lattice Boltzmann equation [18]. The defects in the application should be constantly overcome to make the results obtained in the application more accurate and can obviously guide the production practice.

Expand mine application areas and improve simulation accuracy

The application of the current two-equation model in the mine mostly focuses on the simulation of dust and gas in different ventilation systems and methods, and there is still a certain gap between the calculated accuracy and the measured value. At present, the research on gas and dust in the relevant diverticulum, goaf, transport lanes and other areas and local ventilation methods are still relatively few, and have not yet formed more significant results. Therefore, it should be the focus of future study. In the numerical simulation experiment, when the flow field is numerically simulated by the k-ε model, the values of the turbulent energy k and the turbulent energy dissipation rate ε have an important influence on the flow field solution results [19]. The k and ε values obtained by different turbulent kinetic energy and turbulent energy dissipation rate calculation methods are different, and the obtained solution accuracy is different. Therefore, the simulation results should be continuously improved compared with the measured values or experimental results. Improve the accuracy of analog calculations.

References

[1]. Zhang Zhaoshun. Turbulence [M]. Beijing: National Defense Industry Press, 2002: 7.
[2]. Lu Ming, Sun Xihuan, Li Yanjun, et al. Turbulence numerical simulation method and its characteristic analysis [J]. Journal of Hebei Institute of Architectural Science and Technology, 2006, 23(2):106-110.
[3]. Zheng Sheng. Research on numerical simulation of regularities of dust distribution of front pumping back pressing hybrid ventilation in machine operating work face [D]. Taiyuan: Taiyuan University of Technology, 2014: 27.
[4]. Zhao Xuefeng, Mao Zeyu. Research progress on numerical simulation methods of turbulent flow [C]. National Structure Engineering Conference. 2008: 604-607.
[5]. Zhu Ligang. Progress in numerical simulation of turbulence [J]. Journal of science and technology information in Jiangsu, 2011(5): 38-39.
[6]. Zhu Ligang. Numerical simulation of the flow around a multi barrier on the wall of a channel [D]. Nanjing: Nanjing University of Science and Technology, 2007:24.
[7]. Tang Minbo. Numerical simulation of flow field of pressurized draft airflow in driving face [D]. Nanchang: Jiangxi University of Science and Technology, 2013: 27-96.
[8]. Yao Haifei,Deng Zhigang, Li Jiliang, et al. Numerical simulation for airflow movement and dust transportation of exhaust ventilation [J]. Coal Mining Technology, 2014(5): 96-99.
[9]. Zhang Jianguo, Zhang Wenyu, Jin Chao, et al. Numerical simulation of dust distribution regularities in fully mechanized rock tunneling face with far pressing near absorption ventilation system. [J]. Coal Engineering, 2017, 49(2): 93-96.
[10]. Wang Weijian. Numerical simulation and experimental research of front pumping back pressing hybrid ventilation in machine operating work face [D]. Xiangtan: Hunan University of Science and Technology, 2012: 26-71.
[11]. Feng Jinpu. Research on flow field and gas distribution in gob of Y type ventilation mining working face [D]. Jiaozuo: Henan Polytechnic University, 2012: 11-52.
[12]. Qi Liangfeng. Research on gas field in gob with Y type ventilation system by numerical simulation [D]. Huinan: AnHui University of Science and Technology, 2009: 5-69.
[13]. Wang Kai, Wu Weiyang. Numerical simulation of flow field and gas transportation in goaf of fully mechanized sub level caving face with shape ventilation system [J]. Journal of China University of Mining & Technology, 2007, 36(3): 277-282.
[14]. Gao Jianliang, Zhang Yanli. The influence of the location of level 1 grid nodes on the result of local ventilation flow field calculation [J]. Journal of Safety and Environment, 2010, 10(5): 150-152.
[15]. Wang Haiqiao, Shi Shiliang, Liu Ronghua, et al. Numerical simulation study on ventilation flow field of wall-attached jet in heading face [J]. Journal of China Coal Society, 2004, 29(4): 425-428.

[16]. Liu Ronghua, Wang Pengfi, Song Jia, et al. Numerical research on flow field characteristics of rotating jet shielding ventilation in working face [J]. Journal of Safety and Environment, 2012,12(6):198-203.

[17]. Huang Sheng, Wang Chao, Wang Shiyang. Application and comparison of different turbulence models in the computation of a propeller's hydrodynamic performance [J]. Journal of Harbin Engineering University, 2009, 30(5): 481-485.

[18]. Cui Guixiang, Xu Chunxiao, Zhang Zhaoshun. Progress in numerical simulation of turbulent large eddy [J]. ACTA Aerodynamica Sinica, 2004, 22(2): 121-129.

[19]. Gao Jianliang, Liu Jinjin. Research on calculation methods of simulation calculating the turbulence model in auxiliary ventilation field [J]. Journal of Henan Polytechnic University (Natural Science), 2011, 30(5): 510-515.