First International Symposium on Mine Safety Science and Engineering

Comparative Research on FLUENT and FDS’s Numerical Simulation of Smoke Spread in Subway Platform Fire

Wang Binbin*

Department of Fire Protection Engineering, Chinese People's Armed Police Force Academy, Langfang65000, China

Abstract

This paper describes the methods for simulation of the fire smoke in domestic and foreign subways. By using two simulation software FLUENT and FDS, comparative study of their mathematical models was conducted. By comparison between simulation software field and simulation technology theory as well as the contrast of the physical model of fire smoke spread, and according to the smoke spread test results in Tianjin Xiawafang Subway Station, this paper compared the numerical results of the two simulation software respectively, expecting to use for reference for the study of smoke spread in subway platform fire in the future.

Keywords: subway fire; smoke control; FLUENT simulation; FDS simulation

Subway provides a convenient means of transportation for urban residents, but also brings a new project for fire protection. Because of the dense flow in subway station and the difficulty to evacuate underground, a fire once happened may result in serious consequences. Moreover, smoke in subway fire is one of the main causes of death. Subway fire happened successively is a wake-up call [1]. Therefore, it is a serious problem for the subway transportation system to enhance safety awareness and take preventive measures.

At present, the research method of subway fire at home and abroad mainly contains three types: substance experiment research, reduced scale experiment research and computer simulation research. Comparatively speaking, computer simulation research gradually has advantages with its low cost, fast speed and convenience. This paper compares and analyzes the dissimilarities, advantages and

* Corresponding author. Tel.: 0086-13363606301
E-mail address: srzhai@163.com

© 2011 Published by Elsevier Ltd. Open access under CC BY-NC-ND license.
disadvantages of FLUENT and FDS simulation software in the respect of mathematical model, simulation field, field simulation technology theory and its physical model, then simulates train fire respectively in Tianjin Xiawafang subway. On the basis of smoke spread test results in Tianjin Xiawafang subway, this paper compares and analyzes the simulation result of two simulation software respectively.

1 The comparative study of FLUENT and FDS simulation software

1.1 The comparison of mathematical model

Basic conservation equation

With regard to field simulation, there are six physical parameters to mainly describe flow field, they are as follows: Velocity component in the three coordinate directions: \( u \), \( v \), \( w \), parameter of temperature field \( T \), concentration of smoke \( C \) and pressure of flow field \( P \). The flow of smoke follows the law of conservation of mass, namely, energy equation, momentum equation, equation of continuity and composition equation [2].

(1) Continuity equation:

\[
\frac{\partial \ell}{\partial t} + \frac{\partial}{\partial x_i}(\ell v_i) = 0
\]  

(2) Energy equation:

\[
\frac{\partial \ell}{\partial t}(\ell c_p T) + \frac{\partial}{\partial x_i}(\ell v_i c_p T) = \frac{\partial}{\partial x_i}(\lambda \frac{\partial T}{\partial x_i}) + q_r - q_s
\]

In the equation: \( c_p \) is gas heat capacity at constant pressure, \( \lambda \) is gas thermal conductivity, source terms \( q_r \), \( q_s \), respectively expressed radiant heat term and combustion heat source item.

(3)Momentum equation:

\[
\frac{\partial}{\partial t}(\ell v_i) + \frac{\partial}{\partial x_j}(\ell v_i v_j) - \frac{\partial \ell}{\partial x_i} + \frac{\partial}{\partial x_i}(\ell v_i c_p T) = \left[ \mu(\frac{\partial v_i}{\partial x_j}) + (\frac{\partial v_i}{\partial x_j}) \right] - \frac{2}{3} \frac{\partial}{\partial x_i}(\mu \frac{\partial v_i}{\partial x_i}) + \ell g_j
\]

(4)Mass equation:

\[
\frac{\partial}{\partial t}(\ell c_s) + \frac{\partial}{\partial x_i}(\ell v_i c_s) = \frac{\partial}{\partial x_i}(D \frac{\partial c_s}{\partial x_i}) - W_s
\]

In the equation: \( c_s \) expressed \( s \)-component gas mass fraction, \( D \) expressed \( s \)-component gas diffusion coefficient, \( W_s \) expressed \( s \) components rate of gas in the combustion process of the chemical reaction.

The equation model of FLUENT software

With FLUENT software, this paper employs standard k-\( \varepsilon \) two-equation turbulence model, which is widely used in engineering, and makes three-dimensional numerical simulation of the platform space flow field under fire conditions. During the simulation, SIMPLEC algorithm is employed to solve Reynolds time-averaged N-S equation. Differential equation of field simulation to control fire process can be written in the following general form [3]:

\[
\frac{\partial(\ell \phi)}{\partial t} + \text{div}(\ell \mu \phi) = \text{div}(\Gamma \text{grad} \phi) + S_\phi
\]
In the equation, four items are time item, convection item, spread item and source item. \( \phi \) is generic variable. \( \Gamma \) is diffusion coefficient.

The equation model of FDS software

With FDS software, this paper applies large eddy simulation (LES) numerical method to solve low-speed, heat-driven-flow Navier Stoke equation. It focuses on smoke and heat transfer calculations in the fire hazard, mainly describes the turbulent mixing of gas fuel and combustion product with surrounding air, and its main idea is to make gas vertex of mixing big enough to provide enough accurate calculation results for fluid dynamics equation. Large eddy simulation technology not only overcomes the shortcomings of poor accuracy brought by only getting the average of the flow field but not reflecting chronological characteristics of flow, but also reduces the enormous computational simulation by the relatively coarse mesh [4]. It divides the building space into multiple small grids, solves every conservation equation by numerical method, and can accurately anticipate physical data of fire such as fire pressure, temperature, speed and flow of smoke. Large eddy simulation of the basic equation of fire dynamics is the simplified low Mach number flow equation [5], and is shown in common coordinate system as follows:

Mass conservation equation:

\[
\frac{\partial \rho}{\partial t} + \rho \nabla \cdot \mathbf{u} = 0
\]  

Mass conservation equation:

\[
\frac{\partial}{\partial t} (\rho Y_i) + \nabla \cdot (\rho \mathbf{u} Y_i) = \nabla \cdot (\rho D_i \nabla Y_i) + W_i
\]  

Momentum conservation equation:

\[
\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot (\tau) + \rho g + \mathbf{F}
\]

Energy conservation equation:

\[
\frac{\partial}{\partial t} (\rho h) + \nabla \cdot (\rho h \mathbf{u}) = \frac{\partial}{\partial t} \left( \rho \frac{h}{c_p} \right) + \nabla \cdot (\rho h \mathbf{u})
\]

Gas state equation:

\[
P_0(t) = \frac{\sum_i Y_i}{M_i}
\]

In the equations 6-10, \( p \) is density of flow field; \( \mu \) is velocity of flow field; \( p \) is pressure of flow field; \( h \) is total enthalpy of flow field; \( T \) is temperature of flow field; \( Y_i \) is the mass concentration of component \( i \) in flow field; \( D_i \) is the diffusion coefficient of component \( i \); \( W_i \) is the chemical reaction speed of component \( i \); \( M_i \) is Moore molecular weight of component \( i \); \( \tau \) is the viscous stress tensor of speed flow field; \( R \) is the universal gas constant; \( q_r \) is the flux equation of thermal radiation; \( q \) is the heat released by the fluid combustion.

1.2 Comparison in software simulation field

FLUENT software provides abundant physical models, including ideal gas model, actual gas model, combustion model, physical parameter, rotation system model, heat transfer model and specific boundary conditions of external and internal flow. In addition, FLUENT software contains eight frequently-used
turbulence models in engineering (including S-A model of an equation suggested in 1992, two-equation k-ε model, Reynolds stress model and latest LES simulation), and every model contains some sub models. No software except FLUENT software can provide such abundant physical models [6]. Because of abundant physical models, it is used widely, from wing air flow to furnace combustion, from bubble tower to glassmaking, from blood flow to semi conditioner manufacturing, from clean room to the design of sewage disposal plant; moreover, its powerful simulation capability broadens its application in the field of rotating machinery, aerodynamic noise, combustion engine and multiphase flow system.

FDS software, developed by fire prevention lab of American national technical standards agency, is a kind of fire simulation software based on field simulation. This software is the unique professional fire simulation software developed by authority. It has been tested by many living examples since it is put into practice, and has been used widely in fire safety engineering. Compared with other field simulation software, it is more pertinent. Through years of development, FDS works out a lot of fire problems in fire protection engineering meanwhile provides a tool to study basic fire dynamics and combustion science [4].

1.3 Comparison of field simulation technology principle

Field simulation is a more advanced but more complicated method, which needs mass calculation. Computational Fluid Dynamics (CFD) is a subject, which anticipates fluid flow, heat transfer, mass transport, chemical reaction and related phenomenon by solving mathematical equations representing the physical laws.

FLUENT software adopts finite volume method, which divides calculated area into a series of control volume. Every control volume has a node as a representative. Derive the discrete equation by controlling the volume control equation for the integral. During the process of deriving, the function and first derivative of the interface need to be assumed, and this constitute mode is finite volume discrete method. Finite volume method provides three numerical algorithms, while any other commercial CFD software can only provide one of them: Segregated Solver, Coupled Explicit Solver and Coupled Implicit Solver. Grid generator GAMBIT of FLUENT has outstanding unstructured grid generation capacity, and is considered as the classic preprocessor of present commercial CFD software.

FDS software is adopted to anticipate fire environment under simulated and most adverse credible design fire. It provides two numerical simulation methods: DNS and LES, which can be used in three-dimensional simulation of fire circumstances. This model calculates fluid dynamics based on finite element method concept, has abundant related literature information, and has been tested by large-scale and full-scale fire experiment.

1.4 Comparison of other aspects

FLUENT has powerful post processing function to fulfill required function of CFD calculation, including velocity diagram, contour map, contour surface map, flow path chart, and integrating function to work out force, torque and its corresponding coefficient and flux of force and torque. Parameters users cared about and the error of calculation can be tracked and shown dynamically. Except untreated output data, FDS model provides multiple chart output models, which contribute to observe data intuitively. Section file, contour surface, thermocouple and boundary file are all used for this purpose. The graphical display of output data is disposed by a procedure named Smoke View, which is developed to show output data of FDS specially. Section file is colored slice, or the section going through the whole control volume. Through this section, users can observe temperature distribution of gas intuitively, and can observe the temperature distribution and its variation with the change of time. According to the research content of
this paper, section file is used to estimate the temperature of air, visibility and obscuration coefficient [6].

2 The contrast on subway platform fire numerical simulation

2.1 Setting of fire scene

Xiawafang subway station of Tianjin Metro Line 1 is located in Dagu South Road, which in the south of the Ning Bo Road, north of the Qiong zhou Road. And it is the transfer station of 1#, 5# lines. 1# line and 5# lines in the Da gu South Road and Feng hua Road intersection into a "cross" intersection. It is an island platform, which has effective platform center mileage K17 +073, total length 204.3m, effective platform 120 m, four entrances and two ducts.

The fire source setting of the subway platform

In order to determine the fire intensity of the train fire, the Sichuan Fire detection center of the Ministry of Public Security had metro train fire a single full-scale model test. Test data showed: the fire intensity of ordinary single subway train is 2.75MW. The test results assumed the body of trains normal fuel (baggage, train seats and ornaments, etc.) were on fire, did not consider the firing of locomotives. In this article, for convenience of calculation, we follow a fixed fire power calculation, and considering the motorcycle ignition. Therefore, the train fire intensity of numerical simulation will be set to 5MW. The fire point in the middle of the middle compartment, combustion as non-premixed combustion, the fire source material for hydrocarbons substances gasoline, which using the Pre-treatment software prePDF of FLUENT to define the fire source parameters.

The settings of subway station ventilation system

Subway ventilation system is divided into two categories: the first category is a system building by ventilation and exhaust, by the fans, silencers, pipes, and air and wind pavilion. According to the forward or reverse of the fan achieves the air or exhaust of the system. Generally a lot of tunnel ventilation uses such system. The second category is ventilation system and exhaust system set separately, becoming relatively independent of each system. The smoke vents and air vents are located above the carriageway and the site distribution hall at the top. As the subway station underground space is small, expensive, pipeline complex and difficult to set up an independent exhaust system, so the exhaust system and ventilation system ventilation system combined normally. According to the fire position, the ventilation mode is divided into tunnel ventilation mode and site ventilation mode. When the fire occurred in the tunnel, mainly used to exhaust smoke by tunnel ventilation mode; when the fire occurred at the site, mainly used to exhaust smoke by site ventilation mode[7].

In this paper the main ventilation fan is to exhaust smoke with down to sent and up to exhaust . Fire source position shown in Figure 1:
2.2 The basic assumptions of simulation

In the case of the subway station fire, before the simulation of the smoke unsteady flow and heat mass transfer, we may have the next few basic assumptions:

(1) Smoke plume of fire could be seen as the ideal multi-component gas, air flow and smoke plume follow the ideal equation of state.

(2) Fire smoke plume in the flow of the process is no longer a chemical reaction.

(3) As the analog of the subway station is equivalent to a large space simulation, it can be considered the fire source is oxygen-rich combustion, high-temperature gas arising from fire mainly considered components CO2.

(4) The air flow and temperature before fire evenly are a certain value.

2.3 The using of similar criteria

The reference data of this paper came from the 1:5 scale model experiments of Tianjin Xiawafang subway station in the Sichuan Fire Research Institute in November 2006, and this experiment is the largest subway platform model experiments.

In line with the theoretical analysis of the similar models and experimental design part, confirmation of the numerical simulation result from similar model experiment need to meet certain experimental conditions. Model experimental platform is according to similarity law to design. Therefore, in the corresponding boundary conditions, ventilation and fire source-power mode and other conditions, in theory, the simulation results of physical prototypes should be similar to the model results has corresponding correlation[8].

The numerical simulation results of central subway train fire prototype need test and verify on the basis of the experimental results of model experiments. Be noted that, numerical simulation can get a series flow parameters of each simulated space node, such as speed, temperature, concentration, pressure, etc. But in the model test data acquisition, subject to field experiment conditions, we can not get all the flow field parameters of the space. The results of the record mainly include a fixed measuring point of the flue gas temperature and height, the fire point temperature, wind speed and by the entrance of the experimental observation and camera equipment receive the information.

Subject to the constraints of the experimental data record, as following, we compared the temperature and smoke layer thickness of the axis in the site layout of the five measuring points in the numerical simulation for the time 120s and 360s. Table 1 is for the parameters similar relationship.

| Table 1 Model experimental parameters similar relationship |
|----------------------------------------------------------|
| Parameter relationship | Relationship Equation |
| Geometrical relationship | $x_m = x_a \lambda L$ |
| Time relationship | $t_m = t_a \lambda L^{1/2}$ |
| Temperature relationship | $T_{mL} = T_a$ |

Of which $x_m$ is model size, $x_a$ is prototype size. The experimental results of the model experimental platform have a correspondence $t_m = t_a \lambda L^{1/2} = 0.447 t_a$ between the prototype numerical simulation results in time. In the record of the experimental temperature and smoke layer thickness, we did a record per 5s. In comparing the experimental results and simulation results, the time of experimental data are converted to the time of the prototype state.
2.4 Simulation results of FLUENT and FDS

Comparing physical model

FLUENT modeling grid generated model, as Figure 2:

![FLUENT model meshing chart](image)

FDS modeling grid generated model, as Figure 3:

![FDS model meshing chart](image)

FLUENT uses unstructured grid to reduce the time of generating the grid, simplifies the simulation of the geometry and mesh generation process. And compared with the traditional multi-block grid structure, it can simulate the flow field of more complex geometry, and with the characteristics of the grid more adapt to the flow. FLUENT also able to use in the appropriate volume mesh, the grid block structure. In the 2D flow, FLUENT can process quadrilateral mesh and triangle mesh; in the 3D flow, it can handle the tetrahedral mesh, hexagonal mesh, pyramid, grid and wedge-shaped grid (or a mixture of the grid). These flexibility characteristics of the grid enable us to select the grid type, you can determine the most suitable for specific applications of grid topology.

![Six-sided shape meshes](image)  ![Hexahedral / wedge meshes](image)

For FDS based on linear grid to solve solution equations, in the direct modeling, we should pay attention to the construction of a rectangular region to accommodate the physical background of the grid. Because an important part of the calculation use Poisson solver based on the Fast Fourier Transform (FFTs), so The size of the grid must be in the form of $2^l 3^m 5^n$, and $l, m, n$ are integers. Rectangle can only be used to simulate complex shape, the result will have a certain degree of distortion.

In summary, FLUENT has obvious advantages compared with FDS in the mesh, to be more accurate to restore the true face of physical construction.

Simulation screenshots
120s, the temperature distribution along the axle wire face of subway station as Figure 6:

(a) FLUENT simulation
(b) FDS simulation

Figure 6 the temperature distribution along the axle wire face of subway station (120s)

360s, the temperature distribution along the axle wire face of subway station as Figure 7:

(a) FLUENT simulation
(b) FDS simulation

Figure 7 distribution along the axle wire face of subway station (360s)

120s, CO2 concentration distribution along the axle wire face of subway station as Figure 8:

(a) FLUENT simulation
(b) FDS simulation

Figure 8 CO2 concentration distribution along the axle wire face of subway station (120s)

360s, CO2 concentration distribution along the axle wire face of subway station as Figure 9:

(a) FLUENT simulation
(b) FDS simulation

Figure 9 CO2 concentration distribution along the axle wire face of subway station (360s)
2.5 Comparison between the simulation and test results

Data comparison 120s, data comparison as follow:

(a) the middle height (120s)

(b) the roof height (120s)

Figure 10 the temperature comparison of the platform different heights (120s)

(a) the CO2 concentration comparison of the platform roof height (120s)

(a) the smoke layer thickness comparison of the platform roof height (120s)

Figure 11 concentration and smoke layer thickness comparison

360s, data comparison as follow:

(a) the middle height (360s)

(b) the roof height (360s)

Figure 12 the temperature comparison of the platform different heights (360s)

(a) the CO2 concentration comparison of the platform roof height (360s)

(b) the smoke layer thickness comparison of the platform roof height (360s)

Figure 13 concentration and the smoke layer thickness comparison
Comparative analysis of the data:

a) Comparison of the temperature
- In the 120s and 360s, the middle height, the FDS simulation software compared with the FLUENT software has the better simulation measured results. From the FLUENT software simulation results can be seen, the simulated temperatures are generally higher than measured temperature. It has a higher temperature simulation results than the FDS, and has greater fluctuations in temperature curve.
- In the 120s and 360s, the roof height at the site, the two software is very similar to the trend of the temperature curve. FLUENT simulation results show that five fixed measuring point temperature is higher than the measured temperature, and it is higher than the FDS simulation results. In comparison, the simulation results of the FDS is closer to measured results.

b) Comparison of smoke layer thickness
- In the 120s, the thickness of smoke near the fire source is thicker than that of away from fire. Location away from the fire, smoke density is zero. The simulation results of two simulation software shows that the fire smoke concentration of close to the location is greater than average measured the thickness.
- In the 120s, the simulation results of FLUENT’s smoke layer thickness show that the two measuring points away from the fire source correspond with measured results, but the curve trend is not the same. The results of the FDS simulation show that the overall trend of the curve with the measured results are relatively similar, but away from the fire source there is smoke layer thickness in two measuring points, indicating that smoke has spread to. In the 360s, two software simulation results with experimental results do not match, but their curve is broadly similar trend. Two software simulation results indicate that near the fire source, the thickness of the smoke simulated is is thicker than the measured thickness, but the ones away from the fire source are not as the same. In comparison, the simulation results of FDS are closer to measured results.

c) Analysis of reasons for differences
- Fire power of the two software are both fixed for the 5MW, which achieve stable power in a short time. During the experiment, fire power is a growth process, which may be the reason that the smoke layer of the points near the fire source for two software simulation results are thicker than the measured. In the FLUENT software simulation process, the fire source intensity is indeed the standard, but in the FDS simulation process, its fire did not reach the highest intensity of 5MW in the entire simulation period. This may be several reasons which make the simulation results of FLUENT software generally higher than FDS software simulation results.
- In the experiment, the platform and station hall are both there. The two softwares only simulated platform, the impact of the station hall did not be considered. In the experiment, the spread of smoke is in the station hall, but there will not be simulated this situation. This may result in higher simulation temperature, thicker smoke thickness than the measured temperature.
- The positions of the five measurement points in the FLUENT software are not entirely in accordance with the experiment locations. Because there are three measuring points which are on the stairs, and one which is in the engine room, the stair and room in the modeling have been "digging" from the model body. So simulation positions have been some shift, which generate error. The results of FDS software simulation have some error because that fire intensity of FDS model does not achieve the desired set of standards, or mesh near the fire source is not enough accurate. This may be the reason that the temperature of FLUENT software simulation is higher than results of the measurement and FDS software simulation.
- During the experiment, the smoke leakage phenomenon can be found, after a certain sealing
treatment, the model still can not fully guarantee airtight. So some smoke may be not enter into the hall. This may directly affect the thickness and density of smoke, and also indirectly affect the temperature determination.

3 Conclusions

The train fire smoke spread in Tianjin Xiawafang subway station by using FLUENT and the FDS numerical simulation software. By the simulation results measuring with the measured result, the conclusion is as follows:

- In the simulation field, the FLUENT has more extensive simulation area than the FDS, which has 8 turbulence models to be applied in many fields. And the software of FDS in the field of building fires simulation is more targeted. The FLUENT is more accurate than the FDS in meshing. The FLUENT in modeling, according to the shape of physical building structure, could select the cylinders, rectangular, pentagonal prism and other a variety of shapes to modeling. The FDS can only use the rectangular. In dealing with the completion of simulation data, FLUENT is more convenient than the FDS.
- When the fire occurs in the middle of the train, in 120s and 360s, the simulation results of the FDS are more corresponding with measured results than these of FLUENT in the temperature of middle platform, the roof and the smoke layer thickness.
- For the subway fire smoke spread simulation studies, there is the complexity of the fire source, the diversity of the fire size to need to be further researched. The two simulation software, FLUENT and FDS, the impact for the simulation results are decided by meshing accurate, therefore, the meshing process need more refined, in order to obtain a more accurate simulation results.

Acknowledgements

This research is supported by Distinguishing Specialty Development Project on Fire Protection Engineering Sponsored by the Ministry of Education of China for Colleges and Universities(TS10113).The authors would like to express their gratitude to these foundations.

References

[1] Lu W, Hu Q T, Zhong X X. et al. Gradual Self-Activation Reaction Theory of Spontaneous Combustion of Coal. Journal of China University of Mining & Technology. 2007, 36(1): 111~115. (in Chinese)
[2] Wang H, Wen H, Ge L M. The design and implement of automatic control system for large-scale coal spontaneous combustion experiment unit. Journal of Xi an University of Science and Technology, 2008, 28(1):6~10. (in Chinese)
[3] Deng J, Xu J C, Ruan G Q, et al. Review of the prediction and forecasting techniques of coal self-heating both at home and abroad. Journal of Xi'an Mining Institute, 1999, 19(4): 293-297, 337. (in Chinese)
[4] Xu J C, Wen H, Gou X M. On index of self combustion tendency of coal by spontaneous combustion experiment. Journal of xi’an mining institute, 1997, 17(2):103~107, 126. (in Chinese)
[5] Fire Dynamics Simulator(Version4)-Technical Reference Guide[M]. NIST. 2005 Edition.
[6] Xiao Zenan, Xie Dayong, Sun Xuan. The application of CFD technique used in modeling smoke movement in large space [J]. Fire Science and Technology, 2005. (2): 33-36.
[7] Cai Yun, Li Lidan. The Analysis of the Circulates Mode of Ventilation System in the Case of Fire of Subway [J]. Journal of the Chinese People’s Armed Police Force Academy, 2007, 23(2): 28-31.
[8] Liu Songtao. Subway fire smoke control mode study based on CFD technology [D]. Beijing: Chinese Mining Industry University, 2007.