On Simulation and Experiment of Flow Field of Combustion Coal Fallout Detection Instrument Based on CFD

Yaoshuo Sang 1,2, Qunnan Yan 1, Yong Liu 1, Long Zhang 1, Zhigang Li 1, Chao Ye 1 and Chaohao Guo 1 *

1 Hefei Institutes of Physical Science, Chinese Academy of Sciences, Hefei 230031, China.
2 University of Science and Technology of China, Hefei 230026, China/P. R. China.
Email: 530108687@qq.com

Abstract. In order to further improve accuracy and stability of detection of combustion coal fallout propensity of cigarettes, author of the paper adopted computational fluid dynamics (CFD) technology for a three-dimensional numerical simulation of exhaust system of detection instrument, aiming to study characteristics of flow field near cigarettes. Moreover, a simulation model of eight-channel exhaust enclosure was established, obtaining vector diagram for flow velocity of flow field, velocity contour diagram, and pressure distribution cloud diagram. According to findings, flow field of eight channels is evenly distributed, with slow flow velocity around the instrument but furious inside channels. The wind velocity of cigarette monitoring channel is stable at about 200mm/s specified as per standard. However, there is significant change in pressure and flow velocity at the corners of channels, causing local turbulence. In experiments, average wind velocity of 8 monitoring channels was measured, and simulation results were compared with experiment data. Eventually, a conclusion is drawn that simulation result at cigarette monitoring channels changes consistently with the experimental data, with small errors as a whole. Therefore, the designed exhaust system complies with regulations on wind velocity stipulated by YC/T558-2018 Cigarettes—Determination of Combustion Coal Fallout Propensity of Burning Cigarettes. In a word, this paper is hoped to provide technical support for analogue simulation of exhaust system of cigarette detection instrument, and improve detection accuracy.

Keywords. Combustion Coal Fallout, CFD Analysis, Exhaust System, Experimental Verification

1. Introduction
Combustion coal fallout means that combustion coal falls off or is obviously skewed of cigarettes during smoking or pop-up soot, which will cause smoking interruption, with fire hazards. Moreover, it shall reduce consumer recognition of cigarette brands [1]. Existing researches on combustion coal have made achievements. Specifically, Mullin et al. [2] study the relationship between morphological parameters of combustion coal and combustion velocity, finding that free combustion velocity is positively correlated with cone length and area. LI Bin et al. [3] investigate and analyze behaviors of combustion coal, and determine a method for detecting fallout propensity. Additionally, mathematical regression equations are adopted by QIAO Yuemei et al. [4] to explore how combustion coal fallout is affected by parameters of cigarette paper. They point out in findings that faster combustion velocity of cigarettes requires higher for ash-wrapping effect of cigarette papers. Faster combustion velocity will result in “ashing burst” and “butt falling”. According to standard requirements in YC/T 558-2018
Determination of Combustion Coal Fallout Propensity of Burning Cigarettes, the wind field of detection instrument should meet requirements of Appendix A of GB/T16450-2004 Routine Analytical Cigarette Smoking Machine- Definitions and Standard Conditions [5]: In the instrument, wind velocity of cigarette airflow was maintained at 200±30 mm/s. Wind velocity was the key factor affecting measurement results of combustion coal, so that when airflow moved too fast, free combustion of cigarettes would accelerate. There was a positive correlation between velocity and shape of combustion coal, which enlarged length and area of combustion coal, and improved probability of fallout, affecting detection accuracy of instrument. Therefore, in this paper, the developed CFP800A combustion coal propensity detection instrument was analysed in accordance with YC/T 558-2018 Determination of Combustion Coal Fallout Propensity of Burning Cigarettes. In addition, CFD technology was introduced to explore internal flow field of the instrument, so as to keep airflow velocity around the cigarette close to standard value. This is of great significance to improve detection accuracy and stability of the instrument.

In recent years, CFD (Computer Fluid Dynamics) technology is preferred by scholars to conduct a series of researches on flow field simulation of channels. Particularly, a 20-channel linear smoking machine model is designed by YAN Lihong et al. [6] to measured velocity of cigarettes at different positions; a rectifying net is installed to reduce wind velocity difference at starting position of the combustion. Constructing a negative pressure channel model of plug assembler, XU Jing et al. [7] analyze flow velocity, pressure and turbulence dissipation in negative pressure channel, and optimize negative pressure channel at entrance to improve the fluidity of airflow. Subsequently, scholars YANG Zhanping et al. [8] conduct a numerical simulation on airflow field distribution in wire nozzle, concluding that the larger the gap between inner and outer jackets, the more significant the asymmetrical distribution of inner airflow field. Internal flow field is dried by electrical heating in research of LIU Daoqi et al. [9], who aim to improve uniform flow field by adjusting inlet size, wind pressure and adding a wind baffle. At the same time, k-epsilon turbulence model is applied by Singh J et al. [10] to explore erosion and wear of pipeline by solid-liquid suspended solids, and analysed how erosion and wear are influenced by velocity, concentration, and size. Jeevahan Jeya et al. [11] propose three models - smooth channel, straight channel and oblique channel to cool turbine impeller, and after CFD analysis, determine optimal flow model under high turbulence. Furthermore, LI Ruping et al. [12] discuss influencing factor of fluid in jet-pipe servo valve, reveal characteristics of flow field under different jet-pipe’s diameters, nozzle gaps and jet-pipe angles, and optimize parameters of jet-pipe and receiver. Most of above researches focus on flow field inside flow channel. However, there are few scholars who investigate overall flow field inside the instrument and the detection of wind velocity around cigarettes. In this paper, the author mainly studied wind field of the detection instrument, and introduced RNG k-ε turbulence model in light of influence of wind velocity on detection accuracy of combustion coal. In the meantime, the eight-channel exhaust flow field model was established to analyse vector diagram of flow velocity and pressure distribution cloud diagram, drawing the conclusion that model is uniformly distributed, with good gas fluidity, and stable wind velocity around the cigarettes. In the end, anemometer and special fixtures were used to carry out measurement experiments of wind velocity. According to results, simulation results are in good consistency with the experimental data.

2. Construction of Flow Field Analysis Model

2.1. Working Principle of Exhaust System

Exhaust system of detection instrument is composed of centrifugal fans, exhaust ducts, exhaust hoods, rotating disk, and cigarettes, as shown in figure 1(a). In working, gas is accelerated by centrifugal fan through high-speed rotating impeller, so that flue gas enters exhaust hood along eight channels, and exhausts by fan through the exhaust duct. According to standards, wind velocity of airflow around the cigarette should be between 170 mm/s~230 mm/s during each pumping. The velocity shall be greatly affected by structure of exhaust hood. For this reason, an eight-channel exhaust structure was designed
in figure 1(b), and each channel corresponded to a cigarette, in order to ensure that wind velocity at each inlet was uniform and stable. The author verified rationality of design of the eight-channel structure and distribution of flow field inside the instrument through simulation analysis and experiments.

2.2. CFD Calculation Theory

The analysis model for flow field of combustion coal fallout was established in accordance with the law of conservation of mass, and the law of conservation of momentum. Temperature change in working medium was ignored; for easy calculation, it was supposed that fluid was incompressible and constant Newtonian fluid, with mass conservation equation as follows:

\[
\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{u}) = 0
\]  

(1)

Wherein: \( \mathbf{u} \) is velocity vector; \( \rho \) is density; \( t \) is time.

The momentum conservation equation is:

\[
\begin{align*}
\frac{\partial (\rho \mathbf{u})}{\partial t} + \text{div}(\rho \mathbf{u} \mathbf{u}) &= -\frac{\partial \rho}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + F_x, \\
\frac{\partial (\rho \mathbf{u})}{\partial t} + \text{div}(\rho \mathbf{u} \mathbf{u}) &= -\frac{\partial \rho}{\partial y} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + F_y, \\
\frac{\partial (\rho \mathbf{u})}{\partial t} + \text{div}(\rho \mathbf{u} \mathbf{u}) &= -\frac{\partial \rho}{\partial z} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + F_z,
\end{align*}
\]  

(2)

Wherein: \( \rho \) is density; \( \tau \) is viscous stress; \( F_i \) is physical strength on micro-elements.

Common turbulence models in projects are Standard k-\( \varepsilon \) model and RNG k-\( \varepsilon \) model. Compared with standard k-\( \varepsilon \) model, the RNG model can better deal with flow field structure with great curvature, and three-dimensional flow field model cannot be calculated precisely by the standard k-\( \varepsilon \) model. Therefore, RNG k-\( \varepsilon \) turbulence model is adopted, with equation as below:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right] + G_k + \rho \varepsilon
\]  

(3)
\[ \frac{\partial (\rho e)}{\partial t} + \frac{\partial (\rho e u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \alpha e \mu_{eff} \frac{\partial e}{\partial x_j} \right] + \frac{C_{k}^{1} \rho e}{k} G_{k} - C_{2} \rho \frac{\varepsilon^{2}}{k} \] (4)

Wherein: \( k \) is turbulent kinetic energy, \( \varepsilon \) is turbulent dissipation rate, and \( G_{k} \) means production term of turbulent kinetic energy \( k \) caused by average velocity gradient. The model constants are obtained according to RNG theory: \( C_{\mu} = 0.0845 \), \( C_{1\varepsilon} = 1.42 \), \( C_{1\varepsilon} = 1.68 \), \( \sigma_k = 1.0 \), \( \sigma_\varepsilon = 1.3 \).

2.3. Construction of Numerical Model

Due to complicated internal structure of the instrument, it cannot be simplified into a two-dimensional mode through analysis and calculation. Therefore, three-dimensional calculations were used for simulation and solution. In order to reduce calculations and improve calculation efficiency, the author deleted smoking and pulling units inside the instrument that had a small impact on the flow field, and removed features such as rounded corners and chamfer. The simplified three-dimensional model is shown in figure 2(a), including three parts: outer cover, exhaust hood, and rotating disk. The box body is closed, and the gap between metal plate and profile refers to air inlet that communicates with atmosphere. It was simplified appropriately based on actual model, and 3 air inlets were established on front panel and both sides, as shown in figure 2(b). Moreover, the Boolean subtraction operation was used to cut the rotating disk and cigarettes, forming an enveloping solid that could realistically simulate airflow distribution around the cigarettes. The model was divided by unstructured tetrahedral grid. The author paid more attention to eight-channel exhaust hood, and grid in this area was encrypted to improve calculation accuracy. An expansion layer was added to the wall of model and flow field boundary was refined. At the same time, overall flow field inside the instrument was meshed, as shown in figure 2(b). A total of 2,648,499 nodes and 1,330,742 units were obtained.

![Figure 2. Meshing of Flow Field.](image)

Set boundary conditions of flow field: the air inlet was connected to atmosphere, and set as pressure inlet; reference value was standard atmospheric pressure. The outlet was set as velocity outlet, and wind velocity at outlet was calculated according to pipe diameter and fan parameters. The outer wall of flow field and other areas were all non-slip walls, with RNG \( k-\varepsilon \) turbulence model selected. The iteration steps of the solver were set to 500 steps, and convergence accuracy was 10^{-3}. After iterative calculation based on 500 steps, the continuity convergence accuracy in residual curve was less than the set 10^{-3}, indicating that the model convergence was reliable. An Example. In this example we can see that there are footnotes after each author name and only 5 addresses; the 6th footnote might say, for example, ‘Author to whom any correspondence should be addressed.’ In addition, acknowledgment of grants or funding, temporary addresses etc. might also be indicated by footnotes.
3. Results and Analysis

3.1. Analysis of Flow Field Velocity

Figure 3 (a) refers to flow velocity vector diagram of eight-channel flow field, and colors and arrows represent size and direction of velocity. The fluid enters the box through three air inlets and is discharged through eight channels. The flow field is evenly distributed as a whole, without obvious fluctuations. The outer cover is far away from the channel in four directions, and the gas flow rate is small. After it enters the channel, flow channel gets narrower and the flow velocity is fast. Figure 3 (b) is a vector diagram of flow velocity near one of the cigarettes. It shows that air velocity around the cigarette is stable, with good gas flow, and evenly distributed flow field. Flow velocity vector diagram in the central area is listed in figure 3 (c). The airflow of each channel converges in central area, and flow velocity increases dramatically. Due to eight-channel structure, flow velocity at arc area between two channels is slow, forming a slit. This signifies that airflow is drawn out immediately by the fan, and the flow field is reasonably distributed, no vortex formed here.

![Flow Velocity Vector Diagram](image1)

![Part of the cigarette](image2)

![Part of the slit](image3)

Figure 3. Flow Velocity Vector Diagram.

The x-y section is taken as reference plane to obtain the velocity contour cloud map, as shown in figure 4. Change of wind velocity around the cigarette is observed, and the contour line tells clearly that flow velocity in areas near the cigarette is about 200mm/s. The eight channels correspond to 8 cigarettes, so that flow velocity near the cigarettes is uniform and stable, meeting requirements of GB/T16450. In this section, wind velocity at channel entrance is probably 1800mm/s. As channel cavity becomes larger, flow velocity of gas gradually decreases, and changes uniformly, about 1500mm/s. After gas in eight channels flows into central area, flow velocity will experience collisions with significant fluctuations, causing tremendous energy loss, and decreasing velocity. The flow velocity of eight-channel model will be slightly turbulent inside the channel, but it is more uniform and stable around cigarettes, meeting standard requirements in the range of 200±30 mm/s.

![Analysis of X-Y Section Velocity at Z=300 mm](image4)

Figure 4. Analysis of X-Y Section Velocity at Z=300 mm.
3.2. Analysis of Flow Field Pressure
Figure 5 is cloud diagram of flow field pressure. The pressure is equally distributed as a whole, and judged according to colors. The inlet is atmospheric pressure, and outlet pressure is negative compared with inlet pressure; there is pressure difference at inlet and outlet, which discharges gas inside instrument along channels. The surrounding areas are farther away from suction opening, so the pressure is lower. After gas enters the channel, cavity becomes narrower, volume decreases, and pressure increases. Pressure will experience slight fluctuations at corners of the channels, and reaches the peak at air outlet. The pressure distribution of flow field is consistent with flow velocity that is relatively high in areas with high pressure. It is common to witness gradient change of pressure in eight-channel entrance and the corner inside the channel. The pressure of fluid inside channel tends to increase gradually. The reason is that inclination angle of channels gradually increases when fluid moves along eight channels, and converges to central area, leading to collision that builds up pressure. The pressure of fluid is uniform in outlet pipe area, with stable flow velocity. In summary, the pressure of flow field in eight-channel model is distributed uniformly. Pressure in the channels will increase slightly due to change of shape; pressure of flow field is better, and consistent with flow velocity, satisfying design requirements.

4. Experimental Verification

4.1. Experimental Design
Equipment: self-developed CFP800A combustion coal fallout detection instrument; Schiltknecht-ThermoAir6 Directional anemometer; AHBORN-MA25902A data collector. Since it was necessary to measure wind velocity at outlet by inserting it into the pipe, the HT-9829 thermal anemometer of Laesent was used, with measurement range of 0.1~25 m/s, resolution of 0.01 m/s, and accuracy of ±5%; centrifugal fan was German ebmpapst-D2E146-HR93-01.

Methods: Please refer to figure 6 for experimental instrument and equipment. The wind field test was carried out on the detection instrument, in order to verify whether designed eight-channel exhaust hood met standard that wind velocity near the cigarette should reach 200±30 mm/s. Velocity of eight monitoring channels was measured by Schiltknecht-ThermoAir6 Directional anemometer, and AHBORN-MA25902A data collector. For the purpose of improving measurement accuracy of wind velocity, a special fixture was designed to hold anemometer detector, so as to ensure accurate position of anemometer relative to each measurement point. Anemometer and fixture were introduced in figure 6(b). Eight red marks in figure 6(c) were selected as monitoring channels, and anemometer was adopted to measure average velocity per minute of each point. Each channel was measured eight times.
to reduce experimental errors, followed by a comparison between experimental data and simulation results.

![Anemometer and fixture](image)

(a) Fallout detection instrument  (b) Anemometer and fixture  (c) Monitoring channel

Figure 6. Experimental Instruments and Equipment.

4.2. Data Analysis

Fixture and anemometer were adopted to measure wind velocity of 8 channels, and each channel was measured for 8 times. Measurement results were recorded in Figure 7. According to measurement results, velocity of single channel was uniform and changed slightly. Wind velocity was different between channels, but it was within the deviation range required by the standard. The average wind velocity of 8 channels was 192.3 mm/s. In addition, designed eight-channel exhaust hood and exhaust system were effective to stabilize wind velocity around cigarettes, and reduce influence of wind velocity on fallout detection.

![Velocity measurement of channel](image)

Figure 7. Velocity measurement of channel.

![Comparison of simulation and experiment](image)

Figure 8. Comparison of simulation and experiment.

Average of 8 measurements for each channel was selected, and experimental data was compared with simulation results, obtaining a line chart as shown in Figure 8. There were significant differences between simulation value and experimental value of monitoring channels 5 and 6. The main reason was that when the simulation model was established, smoking unit and lighting unit were deleted for easy calculation, and influence of falling through on wind velocity was ignored, resulting in a slight increase in wind velocity at points 5 and 6. However, experiment and simulation data in the figure changes in a consistent way. In general, simulation of flow field of the detection instrument can
vividly reflect movement process of fluid inside the instrument and in channels, with consistent simulation results and experimental data.

5. Conclusions
In this paper, the author, based on CFD technology, introduces RNG k−ε turbulence model, and establishes simulation model of eight-channel wind field for instrument to detect fallout. At the same time, vector diagram of flow velocity, velocity contour diagram, and pressure distribution cloud diagram inside the instrument are obtained. According to findings, flow field is evenly distributed; gas flowability in eight-channel model is good. Surrounding area is farther from the channels, so, flow velocity is slow, but fast in channel. Wind velocity around the cigarettes fluctuates up and down around standard value.

In accordance with developed combustion coal fallout detection instrument, average velocity of 8 monitoring channels is measured through experiments, and the actual velocity on site meets standard of YC/T 558-2018 Cigarettes—Determination of Combustion Coal Fallout Propensity of Burning Cigarettes. Comparing experimental data with simulation results, the author concludes that change trend of both is consistent, and overall error is small. This verifies correct design of exhaust system, reduces influence of wind velocity on fallout, and improves accuracy and stability of CFP800A fallout detection instrument.

References
[1] YC/T 558-2018 2018 Cigarettes-Determination of Hot Coal Fallout Propensity of Burning Cigarettes.
[2] Mu L, Hui J Q, Tao S C, et al. 2019 Shape parameters of cigarette burning cone and their relationships with burning rate Acta Tabacaria Sinica 25(01) 25-32.
[3] Li B, Gao L, Zhang Y, Hong L, et al. 2018 Observation and measurement of smokers’ ash removal behavior in China Beiträge zur Tabakforschung International/Contributions to Tobacco Research 28(1) 42-50.
[4] Qiao Y M, Zhao Q R, Li L Q, et al. 2017 Analysis of the influence of cigarette paper parameters on cigarette discharge rate Farm Products Processing 23 44-8.
[5] GB/T 16450-2004 2004 Routine Analytical Cigarette-Smoking Machine-Definitions and Standard Conditions.
[6] Yan L H, Zhao Z H, et al. 2015 Improvement of air-flow control system in enclosure of linear smoking machine and its application Acta Tabacaria Sinica 21(03) 25-33.
[7] Xu J, Zhou C J, et al. 2018 Flow field analysis of negative pressure passages for filter assembling in ZJ119-MAX Tobacco Science & Technology 51(09) 79-84.
[8] Yang Z P, Hao X B, et al. 2019 Numerical simulation of air flow field inside tow transport jet in filter rod maker Tobacco Science & Technology 52(07) 95-100.
[9] Liu D Q, Fan C H, et al. 2017 Analysis of internal flow field and optimization of mechanism parameters of negative pressure electric heating dryer based on fluent Food & Machinery 33(09) 94-98+191.
[10] Singh J, Kumar S, Singh J P, et al. 2017 CFD modeling of erosion wear in pipe bend for the flow of bottom ash suspension Particulate Science and Technology 275-285.
[11] Jeevahan J, Durai Raj R B, Mageshwaran G, Sriram V, Joseph G B and Poovannan A 2019 Design and analysis of internal cooling passage of gas turbine using computational fluid dynamics International Journal of Ambient Energy 40(1) 105-109.
[12] Li R P, Nie S L, et al. 2011 Simulation investigation on fluid characteristics of jet pipe water hydraulic servo valve based on CFD Journal of Shanghai University (English Edition) 15(03) 201-206.