OPTIMAL DESIGN OF NOZZLE FOR FILTER BAG CLEANING DEVICE OF COMBINED AIR-CONDITIONING BASED ON FLUID MECHANICS

by

Dongping HUANG**, *Yingke ZHUANG, and Shaojie LIN
Hainan Hongta Limited Liability Company, Haikou, China

Original scientific paper
https://doi.org/10.2298/TSCI190705027H

Nozzles are terminal accessories and important parts of equipment for spraying high speed water flow. The shape of nozzle is the main reason that affects the change of pressure and velocity field. For this reason, this paper uses ANSYS software to monitor the pressure and velocity distribution of the internal flow field of nozzle and the high-speed flow field outside the nozzle. In the paper are compared and analyzed the flow field changes due to different nozzle shapes and the influences of various parameters on the nozzle flow field, providing a reference for the rational use of the structural parameters of nozzle.

Key words: *fluid mechanics, cleaning device, filter bag, nozzle, ANSYS, flow field, modeling, simulation, parameter optimization*

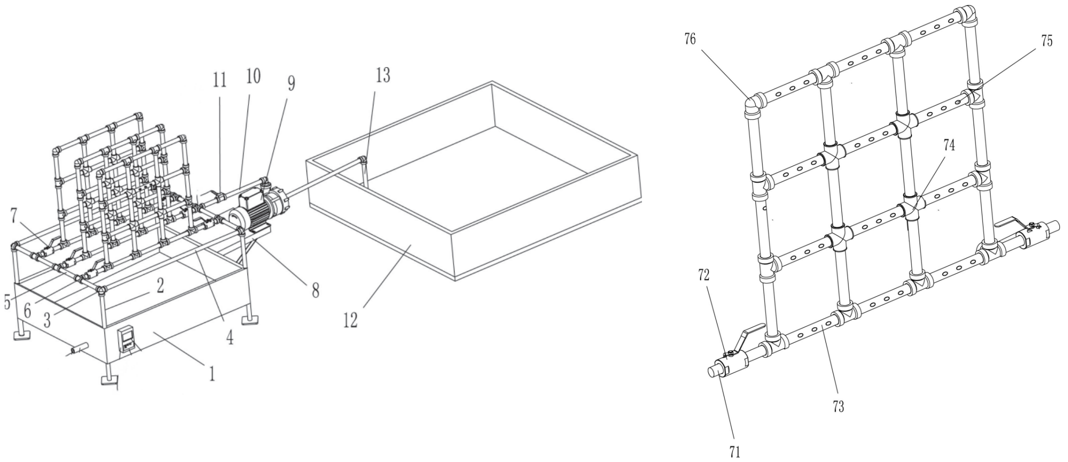
Introduction

The intermediate filter bag is mainly used for intermediate filtration of central air-conditioning and ventilation system, which is an important link of ventilation and filtration of room purification system. The cleanliness degree of the intermediate filter bag seriously affects the air quality of the ventilation system, so it is essential to clean the intermediate filter bag. In this paper, by studying the motion law of hydrodynamics and using ANSYS software to monitor the internal flow field of nozzle and nozzle and the pressure and velocity distribution of high-speed flow field outside the nozzle [1], fluid mechanics analysis is applied to the air-conditioning effect in the design of the filter bag cleaning mechanism, the movement rule of water jet of air-conditioning in the work of cleaning the filter bag is very important, and in cleaning equipment, the speed of water jet and depends on the formation of water jet flow velocity, therefore, for the effect of the filter bag cleaning device in air-conditioning, water jet cleaning device for high and low change rule is to analyze the jet speed motion law and air-conditioning in the effect of the dirt on the filter bag to remove the clean degree of the starting point and the key. The automatic cleaning device of air conditioner filter bag can greatly improve the traditional cleaning method.

Theoretical analysis and derivation

The schematic diagram of the air-conditioning filter bag cleaning device is shown in fig. 1. After the water is pressurized by the pump – 9, it enters the jet system of the cleaning device through pipe – 10 and valve – 11, and sprays the high-speed water through the jet hole – 5. The high-speed water will spray on the air-conditioning medium efficiency filter bag. With the impact

* Corresponding author, e-mail: workhard2009110@163.com



(a) (b)

Figure 1. (a) Schematic diagram of medium effect filter bag cleaning device, (b) the internal cleaning mechanism; *fig. 1(a) it includes: 1 – drainage channel, 2 – bracket, 3 – wall corner tee, 4 – long pipe, 5 – short pipe, 6 – internal wire tee, 7 – internal cleaning mechanism, 8 – installation and construction, 9 – pressurized water pump, 10 – straight pipe, 11 – solenoid valve, 12 – water storage tank, 13 – water suction pipe; fig. 1(b): the internal cleaning mechanism includes – 7: 71 – second short pipe, 72 – first ball valve, 73 – eight perforated pipes, 74 – four-way pipe, 75 – nozzle, and 76 – elbow*

of the high-speed water, the dirt on the surface of the medium efficiency filter bag will be washed clean. This device is mainly used in the maintenance of large central air-conditioning ventilation system. The device is improved on the basis of spraying car washing machine, compared with the conventional water jet device [2], has the following characteristics: in the structural design more simple, easy to assemble and manufacture, low material cost, and can be quickly produced in large quantities. The system is stable and less affected by the external environment.

The model

Principle of model establishment

In this paper, the pipeline dynamic equations are used as the model analysis basis for the cleaning device by ANSYS. Finite element method (FEM) is a numerical method for obtaining approximate solutions to engineering problems. Convection of the FEM to analyze the flow field of strength [3], is the use of the weighted residual method, the PDE of continuous function of closed area is divided into many small regions of the rules of the mass conservation equation and momentum conservation equation to solve, to make the pipe close the basic equations of fluid dynamics, thus to evaluate, on the basis of the equation must be used with the turbulence model. The standard k-ε model is a commonly used turbulence model, where k is turbulence kinetic energy and ε is dissipation rate. The commonly used turbulence model equation is [4]:

$$\rho\frac{Dk}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k+G_b-\rho\varepsilon-Y_M \tag{1}$$

$$\rho\frac{D\varepsilon}{Dt}=\frac{\partial}{\partial \mathrm{x}_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{2}$$

where G_k is the generation of turbulent kinetic energy caused by average velocity, G_b – the turbulence caused by buoyancy effect, Y_M – the influence of compressible turbulent expansion on the total dissipation rate. The σ_k, $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$ are constant coefficients, μ_t is turbulence viscosity coefficient, and:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{3}$$

Grid model

In the air-conditioning filter bag cleaning device, a water pump that can provide 0.2 mPa is selected to provide high-speed water jet for the nozzle. The nozzle is a small hole evenly arranged on the pipe surface. After referring to multiple indicators, the best parameters are selected. The size of the pipe with nozzle is shown in fig. 2. The inner diameter of the nozzle d_{r1} is 40 mm. The outer diameter d_{r2} is 44 mm. The nozzle diameter on the surface of the nozzle d_{r3} is 10 mm. The flow area of the pipe section is A_{r1} = 1256 mm. Connect each nozzle to the controllable valve pump through the three-way pipe, four-way pipe and right-angle elbow pipe. The pipe material is ordinary PVC pipe, the diameter of the inlet is 40 mm, the flow rate is set at 650 Lpm, and the pressure of the inlet is set at 0.2 mPa provided by the pump. The nozzle on the surface of the nozzle is the outlet, The outlet is composed of 24 holes with a diameter of 10 mm. As a result of the contact tube outlet for four row symmetrical design. This is for the convenience of numerical analysis. The outlet is composed of 24 holes with a diameter of 10 mm by simple processing. Fluid meshing can take many forms. The liquid model was meshed with a grid size of 1.6 mm in meshing, with a total of 13256 nodes and more than 5400 units. In the pressure-velocity coupling mode, SIMPLE algorithm was adopted, the second order windward scheme was used for differential discretization of momentum equation, and the second order windward scheme was used for turbulence pulsation energy and turbulence dissipation rate. The number of iteration steps is defined as 300. The tube wall and fluid mesh diagram are shown in figs. 2 and 3 [5].

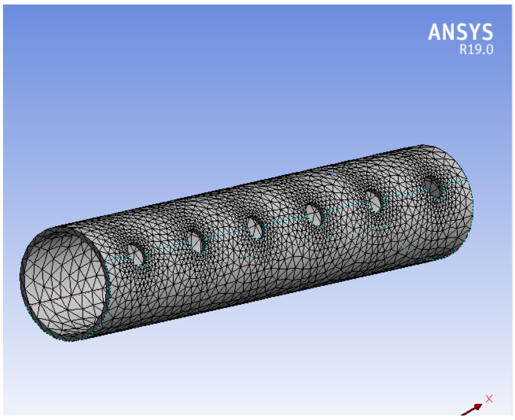


Figure 2. Tube wall grid diagram

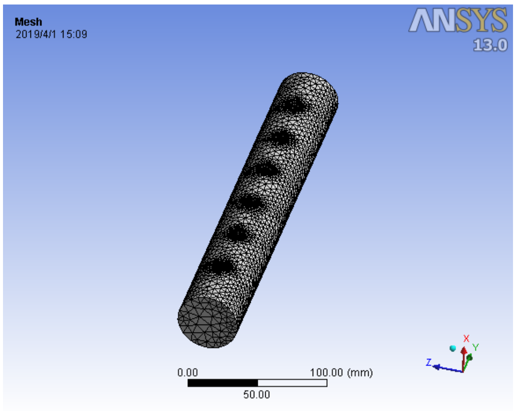


Figure 3. Fluid grid diagram

Define structure and analysis type, add constraint conditions and divide regional boundary in the simulation software, import ANSYS structure file. Good grid was divided by meshing fluid model unit provided import CFX file [6], set the material properties and boundary conditions of fluid model, and the fluid field of the fluid material is set to the water, set up regional exercise way to rest, to define the initial pressure, provided conditions and CFX to solve convergence condition [7], provided by CFX – Solver Manager interface real-time observation of the residual error convergence curves.

The nozzle in the pipeline system firstly adopts the form of uniform and symmetrical orifice nozzle on the surface of the pipeline, so that a 280 mm long common PVC pipe directly becomes the acting surface of the water outlet of the nozzle, as shown in fig. 4:

In the initial condition, on the premise that the pipe wall is a smooth wall, two methods as shown in figs. 4 and 5 are adopted in the calculation of the simulation model: the outlet pressure is set to zero and the inlet velocity is added to one V [ms^{-1}].

$$V_1 = 9\sin\left(\frac{\pi t}{0.04}\right), \quad 0 \le t \le 0.02 \tag{4}$$

$$V_2 = 9, \quad t > 0.02 \tag{5}$$

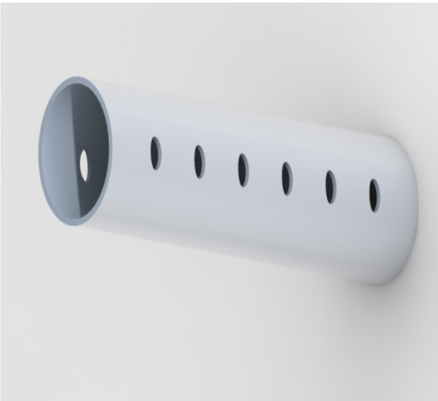


Figure 4, Orifice nozzles on pipe surface

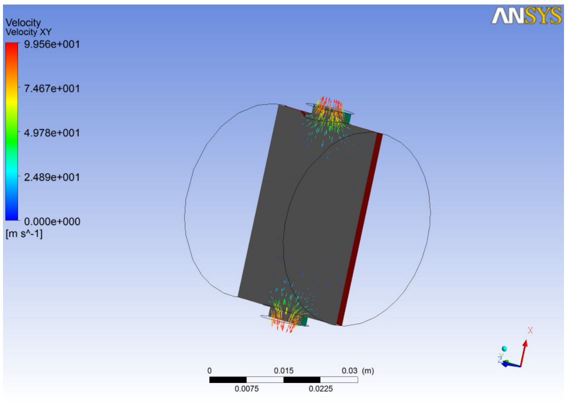


Figure 5. Nozzle velocity distribution

Numerical calculation

As can be seen from fig. 6, in the simulated nozzle flow field, the fluid ejected by the nozzle is conical in shape. With the closer the axial distance from the nozzle, the axial velocity increases. After the high-speed water spurts from the nozzle, it will be affected by air resistance and other factors, and its speed will gradually decrease and the distance will increase to a certain extent, and then the water will run out of energy and fall down.

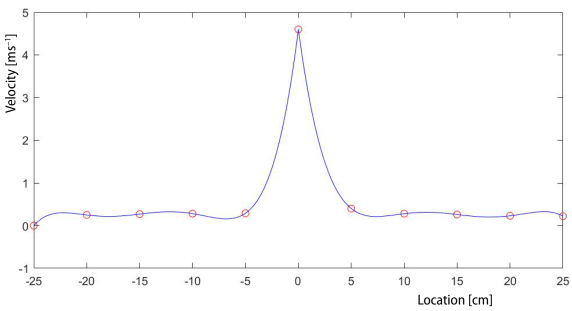


Figure 6. Axial velocity of nozzle flow field

Multi-phase flow model is adopted to simulate the high-speed jet [8], so the process of energy accumulation area of the jet cannot be truly obtained, and only the shape and velocity distribution of the high-speed jet can be simply displayed. The discrete phase model should be used to get a more accurate reaction process in the capacity aggregation area after high-speed jet.

In order to simulate the process of high-speed jet flow by discrete phase model, the continuous flow field must be simulated first. The boundary condition of continuous phase inlet is velocity inlet. The outlet boundary condition is the outlet pressure and is set as the standard atmospheric pressure. After setting the relevant parameters, the operation iteration of the model is started until the model residual curve converges, and the analysis results of the discrete phase model are obtained, as shown in fig. 7. After the establishment of the continuous phase flow field model, the discrete DPM model was added into the high-speed jet field [9], and the computational model was selected k-ε as the model to obtain the simulation analysis results.

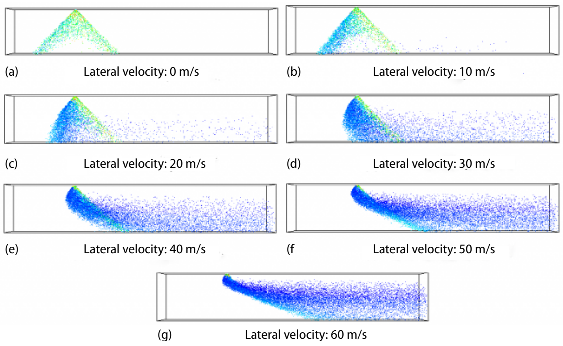


Figure 7. Model of jet dispersion phase

As shown in fig. 7, large droplets of high-speed jet flow are mainly concentrated in the middle region [10-12], which is an energy concentration region, which conforms to the observed jet flow state in practice.

Parameter optimization

The outlet of the original nozzle is designed as a conical nozzle attached to the surface, as shown in fig. 8.

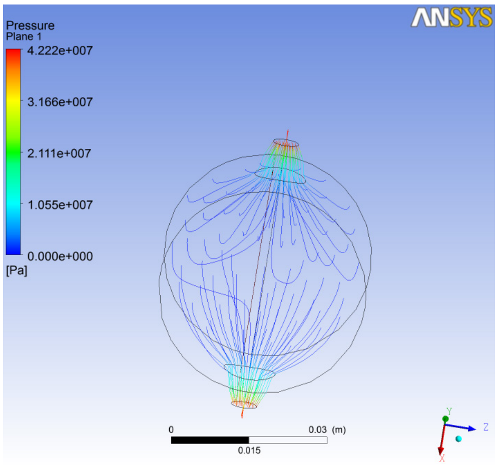


Figure 8. Tapered nozzle

In order to verify the influence of conical nozzle angles at different angles on the velocity, the diameter of the nozzle hole is 30 mm, and the inlet water flow and pressure remain unchanged. In order to verify the validity of the experiment, based on the requirements of the experiment and the experiments done by the predecessors. The cone angles were successively changed to 30°, 35°, 40°, 45°, and 50° and then simulated. After processing the simulation results, the axial high-speed jet velocity is obtained, as shown in fig. 9.

As shown in fig. 9, the speed of high-speed jet flows starts to increase rapidly at around 25 mm, which is mainly caused by conical plane contraction. It can be seen from the picture that when the angle of the cone plane is at α = 30°, α = 35°, and α = 40°, the velocity of the high-speed jet does not have much difference. When the conical plane angle α = 45°, the speed increases significantly. When the angle is reached, the velocity of the high-speed jet stream is lower than the velocity when the angle is reached, so it can be seen that the velocity increases with the angle with the increasing of the angle, the speed increase is no longer obvious, that is, when the angle of the cone plane is α = 45°, the axial velocity of the nozzle reaches the maximum.

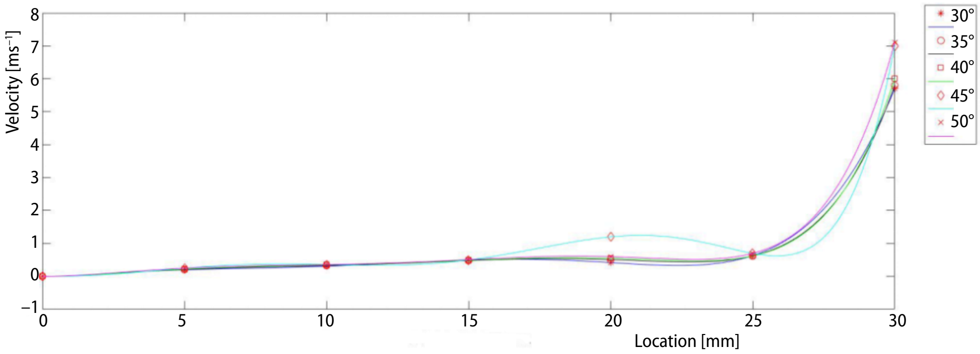


Figure 9. High-speed jet flow diagram with different tapering degrees

Conclusions

Using ANSYS analysis software in the simulated air-conditioning of the nozzle filter bag cleaning device of the internal flow field model, the simulation model, and compares the difference between model flow field under different parameter condition.

- Because the flow in the pipeline will produce turbulence phenomenon, it is necessary to establish a turbulence mathematical model, deduce the turbulence viscosity coefficient according to the turbulence model, and deduce the appropriate nozzle size model parameters according to the coefficient.
- With the common PVC pipe as the model, the pipe and nozzle can be grid more reasonably in CFX.
- Nozzle is the important part of cleaning device, within a certain range to change the shape of the nozzle can change the high speed, the speed of the jet stream, after when the nozzle angle $\alpha = 45°$ of conical surface reaches high speed jet speed increase is not obvious.
- Moreover through research and analysis device of pipe flow field in a 2-D grid of multiphase flow model and discrete model can be intuitive to see the formation process of high speed jet, more intuitive see particle velocity distribution and high-speed jet flow situation, for later in the design of air-conditioning of the pipe cleaning device for filter bag type nozzle system optimization and design provides an important basis.

References

[1] Wang, B., *et al.*, An Analysis of Solid-Fluid Coupling Vibration in Piping by Finite Element Method, *Journal of Harbin Institute Of Technology, 17* (1985), 2, pp. 8-13
[2] Luo, Y., Numerical Simulate of Interrior Flow-Field about Plant Protective Nozzle, thesis, Hunan Agricultural University, Hunan, China, 2008
[3] Wang, W., *et al.*, Impact Analysis of Pigging in Natural Gas Pipeline Tunnel Based on Finite Element Method, *Journal of Dalian University of Technology, 52* (2012), 11, pp. 845-849
[4] Wang, J., *et al.*, Numerical Simulation of Flow Characteristics of Ice Slurry in Vertical Pipelines, *Chinese Journal of Refrigeration, 33* (2012), 2, pp. 42-46
[5] Hao, L., *et al.*, Flow Field Analysis and Parameter Optimization of Atomizing Nozzle Based on ANSYS, *Journal of Agricultural Mechanization Research, 38* (2016), 8, pp. 19-21
[6] Peng, Q., Research on Fluid Flow Characteristics in Pressure Pipeline Based on CFX, *China Equipment Engineering,* (2016), 11, pp. 132-136
[7] Wu, F., Experimental Study on Pipeline Resistance Parameter Detection, *Engineering Technology, 22* (2013), 22, pp. 57-58
[8] Hong S., *et al.*, Research Progress on Fluid Dynamics of Diesel Fuel Injection System, Ph. D. thesis, *Journal of Automotive Engine,* (2015), 4, pp. 1-7

[9] Zhu, H., Wang, H., Computational Fluid Dynamics Analysis of Cavitation Conditions in Water-Cooled Converters, *Power Converter Technology,* (2014), 4, pp. 10-14
[10] Brown, T. S., *et al.*, Analysis of Models for Viscoelastic Wave Propagation, *Applied Mathematics & Nonlinear Sciences, 3* (2018), 1, pp. 55-96
[11] Sudhakar, S., *et al.,* Odd Mean Labeling for Two Star Graph, *Applied Mathematics & Nonlinear Sciences, 2* (2017), 1, pp. 195-200
[12] Hao, L., *et al.,* Flow Field Analysis and Parameter Optimization Based on ANSYS Atomizer, Ph. D. thesis, Hohhot, China Inner Mongolia Agricultural University

Paper submitted: July 5, 2019
Paper revised: September 19, 2019
Paper accepted: September 25, 2019