Research on application of nozzles for ampoule bottle production machine based on ANSYS fluent

Tu Jun*, Pengcheng Xuan
School of Mechanical Electronic & Information Engineering, China University of Mining and Technology, Beijing 100083, China
*Corresponding author e-mail: TuJun0189@163.com

Abstract. In order to determine the suitable combustion temperature suitable for ampoule bottle production, the nozzle and combustion field were numerically simulated and analysed by using ANSYS fluent software to change the inlet diameter and inclination angle of nozzle, respectively. The influence of inlet diameter and inclination angle on the central temperature of combustion flame was determined, and the reference was provided for the condition of ampoule bottle forming. The results show that the inlet diameter has a great influence on the central temperature of the combustion flame and can obtain a more favourable processing temperature range when the inlet diameter is smaller, and the change of the inclination angle has no significant effect on the central temperature of the combustion flame. When the inlet diameter is 0.6 mm and the inclination angle is 20 °, the highest processing temperature range can be obtained. This study can provide a reference for improving the production efficiency of ampoule bottles.

Keywords: nozzle, combustion field, ampoule bottle, numerical simulation.

1. Introduction
Ampoule bottles are currently widely used in the medical field as containers for high-purity chemicals that contain injection preparations and must be insulated from the air. They are mostly made of glass [1].

The production of ampoule bottles is made of finished glass tubes for secondary processing. The key steps are heating, firing, coloring, neck marking, annealing, cooling, etc. The firing molding is the key of the bottle body forming. If the effective flame temperature is not obtained during the sintering process, it is very likely to lead to the deformation of the bottle body or the unqualified size and specification of the bottle body. In the manufacture of ampoule bottles with glass tubes, the nozzle spray flame is needed to heat the softening glass tubes. At present, with the rapid development of computational fluid dynamics (CFD) technology [2], it is possible to use CFD to simulate the dynamic working process of fluid machinery. In actual production, the internal structure of the nozzle will have a great influence on the combustion temperature. Therefore, studying the structure and key influencing factors of nozzles is a major technical problem that technicians in this field need to solve at present.
2. Materials and methods

2.1. Geometric modelling

Figure 1 is a schematic diagram of the nozzle structure. The left end of the nozzle is the intake port, and the different inlet diameters and inclination angles are respectively set. A uniform inlet pressure is set in the numerical simulation process.

![Figure 1. The three-dimensional model of the nozzle.](image)

2.2. Grid partition

In this study, the symmetrical structure of the nozzle is studied, and the simplified model is the plane axisymmetric model. After simplification, the structure of the model is simple, and the meshing of the quadrangle grid in a structured grid can meet the simulation requirements [3], taking the symmetrical distribution of nozzle outlet wide 80mm, long 300mm as the outer combustion flow field region, as shown in Figure 2, the quadrilateral meshes are divided by using Gambit software for nozzles and external combustion flow field. In order to improve the flame effect, the inlet diameter and inclination angle are taken as follows:

(a1): $d = 0.6\text{mm}$, $\beta = 0^\circ$,
(a2): $d = 0.6\text{mm}$, $\beta = 10^\circ$,
(a3): $d = 0.6\text{mm}$, $\beta = 20^\circ$;

(b1): $d = 0.8\text{mm}$, $\beta = 0^\circ$,
(b2): $d = 0.8\text{mm}$, $\beta = 10^\circ$,
(b3): $d = 0.8\text{mm}$, $\beta = 20^\circ$;

(c1): $d = 1\text{mm}$, $\beta = 0^\circ$,
(c1): $d = 1\text{mm}$, $\beta = 10^\circ$,
(c1): $d = 1\text{mm}$, $\beta = 20^\circ$.

The combustion field temperature was studied under the above 9 conditions.

![Figure 2. Nozzle and combustion plane area.](image)

2.3. Boundary condition

In the process of numerical simulation, in order to ensure that the gas does not reflux, the pressure values of the two entrances should be set to the same, and the inlet boundary conditions should be set as pressure-inlet. According to the actual working conditions, given inlet pressure of 3KPa as the research...
condition, the outlet is pressure-out. In order to simulate the real working conditions, the outlet pressure is set at relative atmospheric pressure of 0 Pa, and the temperature is 300K [4].

2.4. Gas fuel pre-treatment
The fuel used in the calculation must be pretreated before the numerical simulation of combustion [5]. The fuel used in this paper is natural gas, its main component is methane, its chemical formula is $CH_4$, and other microelements are not considered [6].

The oxidant is $O_2$ in the air. When the air is selected, only $N_2$ and $O_2$ in the air are taken into account, which account for 79% and 21%, respectively, excluding other trace components.

The chemical reaction formula for methane is as follows:
$$CH_4 + 2O_2 \rightarrow 2H_2O + CO_2$$

2.5. Mathematical model
The fluid flow must follow the law of physical conservation. The basic laws of conservation include the law of conservation of mass, the law of conservation of momentum, and the law of conservation of energy. All flow phenomena in the natural world can be described by two equations: the continuous equation (that is, the mass conservation equation) and the Navies-Stokes equation (that is, the momentum conservation equation) [7].

This paper is studied under the temperature condition of 300K. The transport equation of the turbulent energy in the standard $k-\varepsilon$ model is derived from the exact equation, but the dissipation rate equation is obtained by physical reasoning and mathematical simulation of the similar original equation [9]. The combustion flow is a complete turbulent flow and the molecular viscosity can be ignored, so the turbulence model is chosen as the standard $k-\varepsilon$ model [10]. This model is widely used in practical engineering problems and scientific research.

The standard $k-\varepsilon$ model is a semi empirical model, mainly based on turbulent flow energy $k$ and turbulence energy dissipation rate $\varepsilon$. The turbulent kinetic energy equation $K$ is an exact equation, and the turbulence energy dissipation rate $\varepsilon$ equation is an equation derived from the empirical formula. In order to ensure the closure of the model, the following assumptions are made according to the physical significance [11].

Generate item $= C_{\varepsilon k} \cdot \frac{\varepsilon}{k} \times \text{Turbulent kinetic energy generation term}$

Gradient diffusion term $= \mu_t \cdot \frac{\partial \varepsilon}{\partial x_i}$

Dissipative term $= C_{\varepsilon 2} \cdot \frac{\varepsilon}{k} \times \text{Turbulent kinetic energy dissipation rate}$

2.6. Glass characteristic analysis
Glass is an amorphous material with no fixed melting point and softens at high temperatures. Medical glass materials are generally made of neutral borosilicate glass with better performance. The range of mass fraction of $SiO_2$ is about 75%, $Al_2O_3$ is 5%, $B_2O_3$ is 10%, $CaO$ is 3%, $K_2O$ is 1%, $Na_2O$ is 6% and $Fe_2O_3$ is 0.05%. The flame temperature of secondary reprocessing of glass is 1350°C [12]. Since the production of ampoule bottles is reprocessed by glass tubes, that is, the second reprocessing of glass, the temperature of ampoule processing at 1350°C is taken as the processing temperature of ampoule bottles in this paper, and the temperature distribution of combustion field is analyzed.

3. Results and discussion

3.1. Numerical simulation results of nozzle and combustion field
This study used ANSYS fluent software to perform numerical simulations on 9 experimental models. The results are shown in Figure 3. From the figure, it can be observed that as the flame gradually moves away from the exit, the fuel concentration in the combustion zone gradually becomes thinner, the combustion temperature begins to decrease, the flame propagation speed decreases, the flame thickness
increases, and the flame area gradually expands. The combustion temperature of all combustion fields can reach a maximum of 2180K, but the production and processing temperature of ampoule bottle should reach 1350°C, that is 1623K.

3.2. Numerical comparison and analysis of combustion field

In order to show the effect of different adjusting position on combustion temperature field accurately and intuitively. The simulated data of the central temperature of the combustion field under the condition of the same inclination angle $\alpha$ and different nozzle inlet diameters $d$ are extracted to determine the influence of the nozzle inlet diameter on the combustion temperature in this paper, as shown in figure 4. Comparing the effects of different inlet diameters, it can be found that the smaller the diameter, the easier it is to obtain a higher combustion temperature and the greater the maximum combustion temperature range. And at a position closer to the nozzle, a processing temperature of 1623K can be obtained. Its rationality is inversely related to the size of the inlet diameter.

The simulation data of the center temperature of the combustion field under the conditions of the same nozzle inlet diameter $d$ and different inclination angles $\alpha$ were extracted to determine the influence of the inclination angle of this article on the combustion temperature, as shown in Figure 5. Comparing the effects of different tilt angles, it can be found that the larger the tilt angle, the higher the combustion temperature can be obtained. However, the impact is not obvious.

![Figure 3. Temperature cloud map of combustion center.](image)

![Figure 4. Temperature distribution at different inlet diameters.](image)
Figure 5. Temperature distribution at different inclination angles.

4. Conclusion

(1) The numerical simulation method of CFD was used to simulate the combustion temperature at different inlet inclination angles. It was found that the change of the inlet tilt angle had a certain influence on the combustion temperature. The higher the inclination angle, the higher the combustion temperature, but the effect was not obvious. It reveals the change law of the combustion temperature with the entrance inclination angle.

(2) The numerical simulation method of CFD was used to simulate the combustion temperature of different nozzle inlet diameters. It was found that the change of the inlet diameter had a great influence on the combustion temperature. With other conditions being equal, a more suitable combustion temperature can be obtained with a diameter of 0.6 mm. The maximum processing temperature range can be obtained at position (a). At the same time, the best ampoule production position was determined at 155.182 mm from the outlet of the gas mixing valve.

(3) The research on the nozzle parameters of the ampoules production machine can provide reference for the follow-up engineers and technicians to improve the qualification rate of the ampoules in the nozzle parameters.

References

[1] Jiheng Ding. The development of embedded ampoule bottle liquid agent online detection system. Xi’an University Of Architecture And Technology. 2011:1.

[2] Howard H. Hu. Fluid Mechanics (Fifth Edition) [M]: Elsevier Inc, 2012: 421-425.

[3] Lijuan Wu, Mian Zheng, Caiming Zhang. The Edge Optimization Disposal in Quadrangular Meshes Generation Process[J]. Journal of engineering graphics. 2006,6:49-52.

[4] Jun Li, Qingqing Li, Yuantao Wang, Zhaoli Guo, Xueling Liu. Fundamental flame characteristics of premixed H2-air combustion in a planar porous micro-combustor[J]. Chemical Engineering Journal.2016,283:1190.

[5] Zvonimir Petranovic’, Wilfried Edelbauer, Milan Vujanovic’, Neven Duic´. Modelling of spray and combustion processes by using the Eulerian multiphase approach and detailed chemical kinetics[J]. Fuel,2017,191:28.

[6] Mingkuan Yang. The application of natural gas combustion mode and burner[J].Glass and Enamel. 2001.29(4):31-35.

[7] Pengfei Li, Minyi Xu, Feifei Wang. Proficient in CFD engineering simulation and case practice[M]. Beijing: people post and Telecommunications Press, 2011.10:1-2.

[8] Xueguan Song, Lin Cai, Hua Zhang. ANSYS fluid-solid coupling analysis and engineering example[M]. Beijing: China Water Conservancy and Hydropower Press, 2012:3.

[9] Tianze Cao. Research on Numerical Prediction Technology of Gas Fuel Combustion Flow in Combustion Chamber[D]. Harbin Engineering University. 2012.3:24-25.

[10] Hongjun Zhu. Practical course of FLUENT fluid Analysis and Simulation[M]. Beijing: China
Post and Telecommunications Press, 2010:79-90.

[11] Alan Y. Chen and Lisa Dang. Characterization of Supercritical JP-7’ Heat Transfer and Coking Properties. [C]. AIAA, 2002-0005.

[12] Yingliang Tian. Study on the application of spodumene in neutral borosilicate glass[J]. Glass and Enamel. 2012-8.40(4):11-13.