Research on mixed gas filling process in laser transmission pipeline

Zhenyuan Xu, Wenjing Ma, Junwei Zhang*, Yong Xiang and Liangming Chen

Laser Fusion Research Center, China Academy of Engineering Physics, Mianyang, China

*Corresponding author e-mail: zhangjunwie@caep.cn

Abstract. Aiming at the problem of dynamic displacement of mixed gas in laser transmission pipeline, by selecting a typical section of the pipeline, the problem can be simplified to study the filling of binary mixed gas in a relatively closed cavity with the single entrance and exit. The finite volume method is used to simulate the flow field of this section of pipeline, and the velocity distribution of the binary mixed gas filling process is simulated by using the incompressible N-S equation and the standard k-epsilon turbulence model; the molar concentration distribution of the binary mixed gas over time is simulated. Furthermore, the distribution of flow field and the change of gas composition inside the pipeline were calculated under different inlet velocity conditions. Through the above simulations, the change of flow field distribution in the pipeline with time is described; the formation, development and extinction process of vortex at different time or at different positions in the pipeline are simulated. Based on the works above, the basic research idea and theoretical basis of the study on the mixed gas filling progress of large scale laser transmission pipeline are provided. By comparing and analyzing the characteristics of flow field distribution at different inlet velocity or at different time periods of the same inlet velocity, it will also have certain reference value to guide the implementation and optimization of the gas filling project of laser transmission pipeline.

1. Introduction

In huge laser facility, there are large numbers of laser transmission pipelines which used for high-energy laser transmission, such as long straight square pipe, long straight round pipe, right angle pipe, cross pipe, obtuse pipe, three-branch pipe and so on. The purpose of these laser transmission pipelines is to ensure the safe, stable and efficient transmission of high-energy laser. Therefore, these closed pipelines are often filled with inert mixed gas to ensure the transmission quality of high-energy laser [1, 2]. Generally, oxygen and argon are evenly mixed in accordance with 1:1. Two kinds of energy losses occur when gas flows through these pipes, one is the loss of energy along the way due to the viscosity of the fluid and the friction on the pipe wall; another is the local energy loss caused by local obstacles, such as right-angle areas, bifurcation points of bifurcation pipes and obstacles. Due to the special properties of the fluid itself, the separation and reattachment of the flow will occur when the fluid flows through these local obstacles [3, 4]. The separation of the flow will cause vortex, which will cause pressure drop and energy loss, then it will produce different flow field distribution.
In this paper, a simplified two-dimensional model of the laser transmission pipeline is established by combining the computational fluid dynamics method, and the velocity distribution of the pipe during the filling process of binary mixed gas is numerically simulated by using the incompressible n-s equation and the standard k-ε turbulence model [5]. By simulating the molality distribution of the binary mixed gas changing with time, the detailed flow field distribution in the pipeline is obtained, and the simulation results are analyzed. By comparing the CFD numerical calculation results, the characteristics of flow field distribution in the pipeline under different inlet velocity changes and the same velocity at different periods are studied [6]. It provides a scientific basis for the theoretical research and engineering implementation of gas filling in laser transmission pipeline.

2. Numerical simulation

Since the laser transmission pipeline is spliced together, the most typical section of straight pipeline is selected as the analysis object, and the problem is simplified as filling gas for two-dimensional pipeline. In order to conform to the actual situation, the model of the analysis object is set as a two-dimensional pipe section with a length of 2 meters and a height of 1 meter. The air inlet is located at the upper part of the left wall of the pipe, with a height of 0.1 meters. The air outlet is located at the lower part of the right wall of the pipe, with a height of 0.1 meters. The schematic diagram of the model is shown in Fig 1.

![Figure 1. Schematic diagram of calculation domain.](image)

2.1. The governing equation

The fluid in the calculation area of optical transmission pipeline is incompressible. Ignore the heat transfer and energy exchange between the pipeline and the external environment. The flow state belongs to the turbulence model.

Incompressible n-s equation is used to describe the filling process of mixed gas[7, 8]. In the rectangular coordinate system, the continuity equation and the momentum equation are as follows:

\[
\frac{\partial u_i}{\partial x_j} = 0
\]

\[
\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j} \left( u_j u_i \right) = - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial u_i}{\partial x_j} \right)
\]

Where, \( u \) is the average fluid velocity; \( t \) is time; \( \rho \) is fluid density; \( p \) is the pressure; \( \nu \) is the viscosity coefficient of fluid motion; \( x \) is the coordinate variable; \( i, j \) correspond to the \( x \) and \( y \) directions.

Standard k-ε turbulence model:

\[
\frac{\partial k}{\partial t} + \frac{\partial}{\partial x_i} \left( u_i k \right) = \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_k}{Pr} \right) \frac{\partial k}{\partial x_j} \right] + \frac{\rho_k}{\rho} - \varepsilon
\]
\[
\frac{\partial \varepsilon}{\partial t} + \frac{\partial}{\partial x_j} (\langle u, \varepsilon \rangle) = \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{Pr} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1e} \frac{\varepsilon}{k} \frac{G_j}{\rho} - C_{2e} \frac{\varepsilon^2}{k} \tag{4}
\]

Where, \( k = \frac{u'_j u'_i}{2} \) is turbulent kinetic energy; \( u'_i \) and \( u'_j \) are fluid pulsation velocity; \( \varepsilon \) is the turbulent energy dissipation rate; \( \nu_t = C'_\mu \frac{k^2}{\varepsilon} \) is the turbulent viscosity coefficient; \( G_j = -\rho \frac{\partial u'_j}{\partial x_i} \frac{\partial u_i}{\partial x_j} \) is the turbulent kinetic energy generated by the laminar velocity gradient; \( C_{1e} \) and \( C_{2e} \) are constants.

2.2. The boundary conditions
Referring to figure 1, the boundary conditions are as follows:
1) \( \Gamma_1 \) is the entrance of argon-oxygen mixed gas. Select the speed entry condition. Select 0.5m/s and 1.0m/s as the calculation condition.
2) \( \Gamma_2, \Gamma_3, \Gamma_5 \) and \( \Gamma_6 \) are the surface of the pipeline. Ignore the deformation of laser pipeline, select the non-slip adiabatic wall surface as the boundary conditions.
3) \( \Gamma_3 \) is the outlet of mixed gas, choose the free flow outlet condition.

2.3. Grid processing and computing methods
The whole computing domain is divided by structured grid, and the grid is encrypted at the entrance and exit, shown in Fig 2. Fluent used finite volume method to solve the incompressible n-s equation at low speed [9, 10]. Consider calculation affected by gravity. An unsteady solver is used in iterative calculation, and the iteration time step is 1s.

Figure 2. Geometric model of laser transmission pipeline and computational grid division.

3. Calculation results and analysis
In this part, the flow field is calculated in different flow velocity conditions, the velocity field distribution and the concentration distribution of different gas components are obtained under different time conditions, and the representative results are analyzed in detail.

3.1. The velocity distribution and streamline changed with time
When the inlet velocity \( u = 0.5 \text{m/s} \), the flow field in the pipeline changes with time from the initial stage to the process of stabilization, as shown in Fig 3.
As Fig 3 shown, at the initial stage of filling (t=100s), the mixed gas is injected into the pipeline from the left inlet and then settled to the bottom; Then it gradually diffuses to the right and flows to the outlet. The jet induces the gas in the space on both sides to form a vortex. More vortices (about 3 vortices) are formed in the region on the right side of the flow, while a smaller clockwise vortex is formed in the region on the left. 

When t=200s, the vortex in the region on the left side of the jet increases, while the vortex in the region on the right side gradually fusions from three vortexes into two vortexes, and the mixed gas at the bottom of the pipeline gradually increases.

When t=300s, with the increasing volume of the incident mixture, the pressure and velocity distribution of the flow area in the pipeline continue to change. The space of the left and right regions is basically equal, and a stable clockwise vortex is formed on the left. Due to the compression of the initial space in the right region, two of the original three vortexes gradually degenerate and disappear, and eventually evolve and fuse into one vortex.

When t=500s, the incident mixed gas is gradually stabilized in the upper part of the pipeline, and the area on the right side of the airflow is compressed in a small space. The original vortex gradually rises and shrinks to form a stable counter-clockwise vortex, while the area on the left side of the airflow mainly forms a large clockwise vortex.

When t=1500s, the distribution of flow field in the pipeline is gradually stable over time, and the stable flow field is finally formed as shown in Fig 3 (f).

When the inlet velocity $u=1.0\text{m/s}$, the flow field in the pipeline changes with time from the initial stage to the process of stabilization, as shown in Fig. 4.

As Fig 4 shown, when the incident velocity is doubled, at t=100s, the mixed gas in the pipeline has formed a three-vortex structure. A clockwise vortex is formed when the airflow enters the pipeline from the left inlet and settles to the bottom, while a large and a small vortex is formed in the right area.

When t=200s, the incident gas tends to be stable and concentrated at the upper end of the pipeline. The area on the right side of the airflow is compressed in a small space, and the two original vortexes
degenerate, rise and fuse into a counter-clockwise vortex. The area on the left side of the airflow mainly forms a large clockwise vortex.

When t=1000s, the distribution of flow field in the pipeline gradually becomes stable over time, and finally forms a stable flow field as shown in Fig 4 (d).

3.2. The molar concentration distribution of argon changed with time
When the inlet velocity u=0.5m/s, the flow field in the pipeline changes with time from the initial stage to the process of stabilization, as shown in Fig 5.

![Figure 5. Molar concentration changes of argon over time (u=0.5m/s).](image)

When the inlet velocity u=1.0m/s, the flow field in the pipeline changes with time from the initial stage to the process of stabilization, as shown in Fig 6.

![Figure 6. Molar concentration changes of argon over time (u=1.0m/s).](image)

4. Conclusion
In this paper, the huge laser transmission pipeline mixed gas dynamic displacement problem is simplified to a two-dimensional piping gas chamber inflatable problem. By using Fluent, the change of velocity distribution of gas flow field in the pipeline with time under the conditions of two different inlet velocities was well simulated. The whole process of formation, change and extinction of multi-vortex structure was simulated. At the same time, the change of the molar concentration distribution of argon over time was simulated, and the comparison and analysis were carried out.

1) Mixed gas filling process in laser transmission pipeline is mainly as this: the mixed gas enters the pipeline from the left inlet, after settling to the bottom, it diffuses to the right and flows to the outlet. The jet flow induces the gas in the space on both sides to form a vortex, which changes the pressure and velocity distribution in the pipeline. The molar concentration of argon gas in the pipeline changes with the increase of the injection volume of mixed gas. The concentration of mixed gas on the left side of the jet increases and the pressure increases, so that the jet settling area moves to the right part, finally move
to the right wall. After a period of time, the concentration distribution in the pipeline is stable, and the flow field distribution is stable.

2) The stabilization time of velocity field is earlier than that of concentration field.

3) When the incident flow velocity is 1m/s, the variation trend of the velocity field and argon gas concentration is consistent with the low-speed working condition. However, the initial jet settling position moves to the right and the gas diffusion process is accelerated.

4) Stable time of concentration distribution: The incident flow velocity of 0.5m/s requires 1500s, while the incident flow velocity of 1m/s reaches a stable state less than 1000s. After stabilization, oxygen is mainly concentrated in the middle and upper part of the pipeline (the upper segment has relatively high concentration), while argon gas is mainly concentrated in the lower part of the pipeline (the lower segment has relatively high concentration).

References

[1] Govier, K. Aziz. The Flow of Complex Mixtures in Pipes. Reinhold Van Nostrand Co., 1972.

[2] H. Henry, H. Fauske. Two-Phase Critical Flow of One-Component Mixtures in Nozzles, Orifices and Short Tubes. Trans. ASME, J. Heat Transfer, 95, 179-187, 1971.

[3] A. J. Chorin, J. E. Marsden. A Mathematical Introduction to Fluid Mechanics. Springer, Berlin, Heidelberg, New York, 2000.

[4] Safikhania H, Akhavan-Behabadi, et al. Modeling and multi-objective optimization of square cyclones using CFD and neural networks. Chemical Engineering Research and Design, 2010, 89(3) : 1-9.

[5] Wu C J, Liu D P, Pan J. A study of the aerodynamic and acoustic performance of an indoor unit of a Dc inverter split air-conditioner. Applied Acoustics, 2012, 73(4): 415-422.

[6] J.D. Computational Fluid Dynamics. The basics with application. 2002(5): 150-152P.

[7] J W Dear Doff. A Numerical Study of Three-dimensional Turbulent Channel Flow at Large Reynolds Number. Fluid Mach. 1970(20): 453-480P.

[8] Lin F, Schohl G A. CFD prediction and validation of butterfly valve hydrodynamic force. World Water Congress. 2004.

[9] Versteeg H K, Malalasekera W. An Introduction to Computational Fluid Dynamics the Finite Volume Method. Pearson, Edinburgh England, 2007.

[10] A. Masjedian Jazi, H. Rahimzadeh, Waveform analysis of cavitation in a globe valve. Ultrasonics 49 (2009) 577-582.