Advances in Technology Innovation, vol. 3, no. 3, 2018, pp. 133 - 140

Flowfield Analysis of a Pneumatic Solenoid Valve

Sheam-Chyun Lin[1,*], Yu-Ming Lin[1], Yu-Song Haung[1], Cheng-Liang Yao[2], Bo-Syuan Jian[2]

[1]Department of Mechanical Engineering, National Taiwan University of Science and Technology, Taipei, Taiwan

[2]Metal Industries Research & Development Centre, Kaohsiung, Taiwan, R.O.C.

Received 21 July 2017; received in revised from 24 July 2017; accepted 17 September 2017

Abstract

Pneumatic solenoid valve has been widely used in the vehicle control systems for meeting the rapid-reaction demand triggered by the dynamic conditions encountered during the driving course of vehicle. For ensuring the safety of human being, the reliable and effective solenoid valve is in great demand to shorten the reaction time and thus becomes the topic of this research. This numerical study chooses a commercial 3/2-way solenoid valve as the reference valve for analysing its performance. At first, CFD software Fluent is adopted to simulate the flow field associated with the valve configuration. Then, the comprehensive flow visualization is implemented to identify the locations of adverse flow patterns. Accordingly, it is found that a high-pressure region exists in the zone between the nozzle exit and the top of the iron core. Thereafter, the nozzle diameter and the distance between nozzle and spool are identified as the important design parameters for improving the pressure response characteristics of valve. In conclusion, this work establishes a rigorous and systematic CFD scheme to evaluate the performance of pneumatic solenoid valve.

Keywords: pneumatic solenoid valve, compressible numerical simulation, transient characteristics, pressure-rising process

1. Introduction

Pneumatic system has been used extensively in many areas of industrial applications, such as automation control, medical instruments, and control unit of the vehicle. It is essential to choose an appropriate valve as the interface to electronic controls for performing the required adjustments or actions in accordance to its function design. Among the automobile safety system, an accurate and reliable pneumatic solenoid valve with a short response time is critical in the anti-lock braking system (known as ABS), which offers improved vehicle control and decreases stopping distance. Therefore, the understanding on the flow patterns inside the solenoid valve is in great demand to shorten the reaction time, and thus becomes the goal of this research.

Usually, servo valve and on–off valve are two types of electro-pneumatic valves used in controlling the pneumatic actuator. The expensive servo valves with complex structure are used to achieve the high linear control accuracy. On the other hand, due to the low cost, compact size, and simple structure, the fast-switching on–off valves have received considerable attention in vehicle industry and researchers [1-4]. In 2006, Topçu et al. [5] develops the simple, inexpensive fast-switching valve for applications of pneumatic position control. Four prototype valves have been built and the basic mode of operation confirmed. In addition, the switching characteristics of the on–off valve with 2/2-way function has been investigated both theoretically and experimentally. Simulated results of the valves dynamics were in agreement with the experimental results, and thus the validity of the proposed mathematical model was confirmed.

* Corresponding author. E-mail address: sclynn@mail.ntust.edu.tw
Tel.: +886-2-7376453; Fax: +886-2-27376460

As expected, the magnetic field has a dominant influence on the response characteristics of a solenoid valve. Thus, an optimal design of the magnetic field of a high-speed response solenoid valve is executed by Tao et al. [6]. They used the finite element method to optimize the solenoid valve for achieving larger magnetic force and low power via the changes on parameters and materials. Later, Wang et al. [7] investigated influences of cross-sectional area of the iron core and ampere turn on the static electromagnetic characteristics through numerical simulation. They found that the ampere turn has great effect on electromagnetic force for the magnetic saturation phenomenon. Besides, the simulation method is validated by the experiment.

As regards the flow field analysis, several CFD reports [8-12] are focused on analyzing the flow field inside the valve. Peng et al. [8] adopted the commercial CFD software Fluent to establish CFD model for simulating the inner flow field of a servo valve when the valve spool is located in certain positions. Also, several improvements in the core shape of valve are raised and evaluated via the established numerical model. Later, Ma and Sun [9] used CFD software Fluent to simulate the static and dynamic flow fields of an electromagnetic valve. The function between the mass flow rate and the drop of pressure through the electromagnetic valve was obtained from the results of static flow field simulation. From the numerical simulation of the unsteady flow, the valve closed procedure was calculated. The results indicate that the inner flow field numerical simulation of the valve by Fluent can reflect its working procedure.

More recently, in 2016, Liu et al. [10] conducted a research on a solenoid valve used in the hydraulic control system. Based on the conditions occurring in the operation of the hydraulic drive system, the thermal field of the head is analyzed by ANSYS. It is illustrated that the solenoid valve has a good performance under high temperature condition. They presented a method to monitor the performance of the valve while the reactor is working. From the previous papers, it is demonstrated that numerical simulation can be adopted as a reliable and useful tool in the valve design. Later, Liu et al. [11] presented a nonlinear dynamic model of a large flow solenoid with the multi-physics dynamic simulation software called SimulationX. The dynamic characteristics of this solenoid valve are analyzed and validated by comparing the test and CFD results. In fact, Computational Fluid Dynamics (CFD) is increasingly being used as a reliable method for determining performance characteristics of other valve. Farrell et al. [12] executed a series of CFD investigation on characterizing the opening and closing of check valves. They adopted CFX which is a part of the ANSYS suite of finite element programs, to predict and characterize the performances of swing check and lift check valves. Also, the good agreements are found via comparing the available test data of the modeled valves with the numerical results.

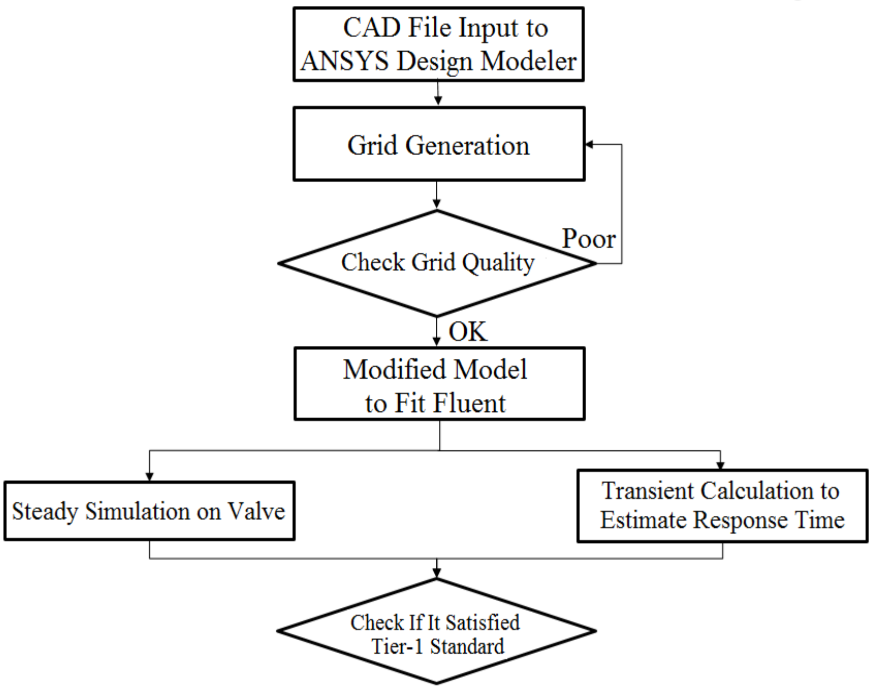


Fig. 1 Methodology of this numerical simulation over a pneumatic solenoid valve

Therefore, this computational fluid dynamics (CFD) study chooses a commercial 3/2-way solenoid valve, which is used extensively in vehicle control system, to examine its dynamic performance. At first, flow-field simulation associated with the valve construction is executed by using the commercial CFD code Ansys Fluent. Then, the comprehensive flow visualization is implemented to identify the locations of adverse flow patterns, which are critical for proposing the improving alternatives. Also, the flowchart for this valve research is illustrated in Fig. 1

2. Working Principle of Charging Process and Description of Physical Model

2.1. Working principle of charging process

Fig. 2 shows the overall valve system, which is composed of the solenoid valve, piston connector, connecting duct, and the outlet vessel. It is necessary to describe the working principle for the pressure-rise process of solenoid valve in brief. For increasing pressure to its setting value for activating the ABS system, a high-pressure (10.1 Bar) air source is connected to the nozzle inside the top portion of valve (see Fig. 3). Thus, this big pressure difference generates a chocking situation (sonic speed at the nozzle exit) and an inflow with the constant mass flow rate at the beginning of this filling process. However, after the vessel pressure reaches a fixed value (near 52.8 % of the source pressure), the flow rate of this inlet airstream becomes smaller with a rising vessel pressure. Finally, this process ends when vessel pressure is equivalent to the pressure source.

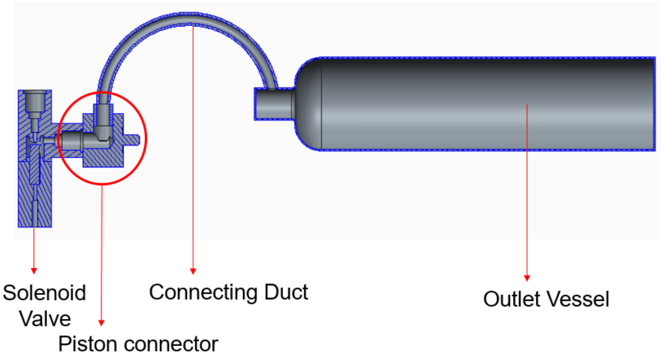


Fig. 2 Overall system of the pneumatic solenoid valve

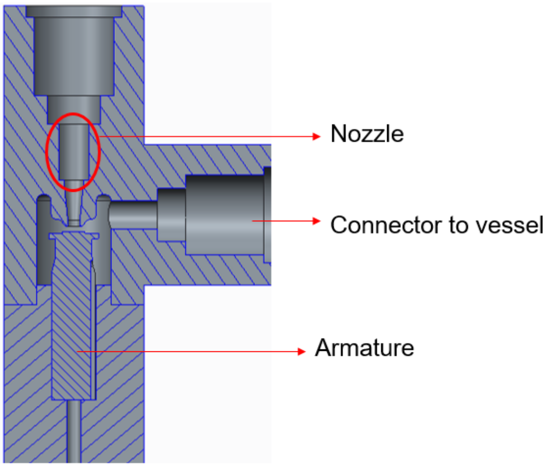
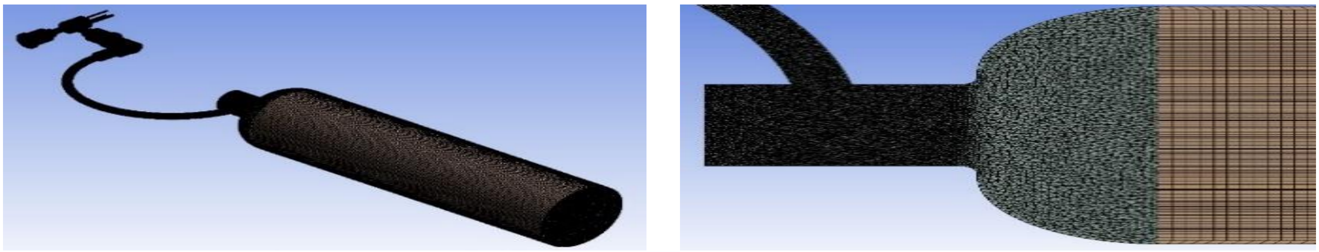


Fig. 3 Open mode of the pneumatic solenoid valve

Clearly, this charging process is an unsteady flow undergoing a significant pressure variation; thus the transient simulation and compressible assumption are needed to realize the complicate physical phenomena. However, it is known that the unsteady CFD simulation not only needs a high-performance server with huge memory resource, but also takes a much longer CPU time to obtain the result. Thus, the steady simulation is carried out on the complete valve system with an opened vessel end in this study for evaluating the flow patterns inside the geometry. Thereafter, a comprehensive analysis of the flow pattern inside of the valve system is executed via the simulation outcomes for finding out the modification possibilities.

2.2. Description of physical model

The actual valve configuration is quiet complex and difficult to establish a numerical model for CFD simulation. Thus, proper simplifications on the CAD file are needed to attain an effective numerical model, which is divided into several portions with different grid densities as indicated in Fig. 4(a). Generally, to capture the actual physical phenomenon precisely, the intense grid distribution is placed on regions with an abrupt property variation on velocity, pressure, or direction. For the valve considered here, as illustrated in Fig. 4(b), these locations include the nozzle, the small clearance between nozzle exit and the armature, expansion part in the connector, and junction between the connecting pipe and the vessel. The total grid number of this numerical model is 8.6 million.



(a) Overall system of the valve (b) Connecting duct to the vessel

Fig. 4 Grid system of the pressure-increasing process for a pneumatic solenoid valve

3. Numerical Scheme

This study simulates the complex flow patterns inside the electromagnetic valve by utilizing the commercial computational fluid dynamics (CFD) software Fluent [13] to solve the fully three-dimensional compressible Navier-Stokes equations with the standard k-ε turbulence model. Also, the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) [14] is implemented to solve the velocity and pressure coupling calculation for steady cases. Hence, the flow visualization inside the valve can be performed and observed carefully to locate the reversed flow patterns.

In this work, several appropriate assumptions and boundary conditions were made to simulate the actual flow patterns inside a ceiling fan. They are described as:

(1) Inlet boundary condition:

The inlet boundary condition of valve is set as P_{abs}=11 bar for serving as an extremely high-pressure input.

(2) Outlet boundary condition

The outlet boundary condition at the right wall of vessel is set as the atmospheric pressure.

(3) Wall boundary condition

This numerical model sets the no-slip boundary condition on the solid surfaces of the solenoid valve system.

All kinds of flowing fluid problems are determined by physical principles, which are expressed in conservative form for mathematical description. They are mass equation (continuity equation) and momentum equation. Moreover, as the fluid is under the turbulent condition, the additional turbulent equation is needed to incorporate with the governing equations. The continuity and momentum equations in conservation form are expressed as follows:

(1) Continuity conservative equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = S_m \tag{1}$$

Here u_i is the velocity, ρ is the density, and S_m is the source term.

(2) Momentum conservative equation

$$\frac{\partial}{\partial t}(\rho u_i)+\nabla\cdot(\rho u_i u_i)=-\nabla p+\nabla\cdot(\bar{\tau}\eta)+\rho g_i+F_i \tag{2}$$

where p is the static pressure, τ_{ij} is the stress tensor, and ρg_i and F_i are gravitational and external body forces, respectively.

Also, the k-ε turbulence model is utilized to solve the Navier-Stokes equations. With respect to the incompressible flow and no source condition under the steady-state, the momentum equation is:

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right]+\frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \tag{3}$$

Note that Eq. (3) is called the Reynolds-averaged Navier-Stokes (RANS) equation, where the Reynolds stress $-\rho u_i' u_j'$ should be appropriately modeled by the Boussinesq hypothesis [15] for relating to the mean velocity gradients. The advantage of this approach is the relatively low computational cost associated with the computation of the turbulent viscosity. The k-ε model computes the turbulent viscosity as a function of turbulence kinetic energy k and turbulence dissipation rate ε:

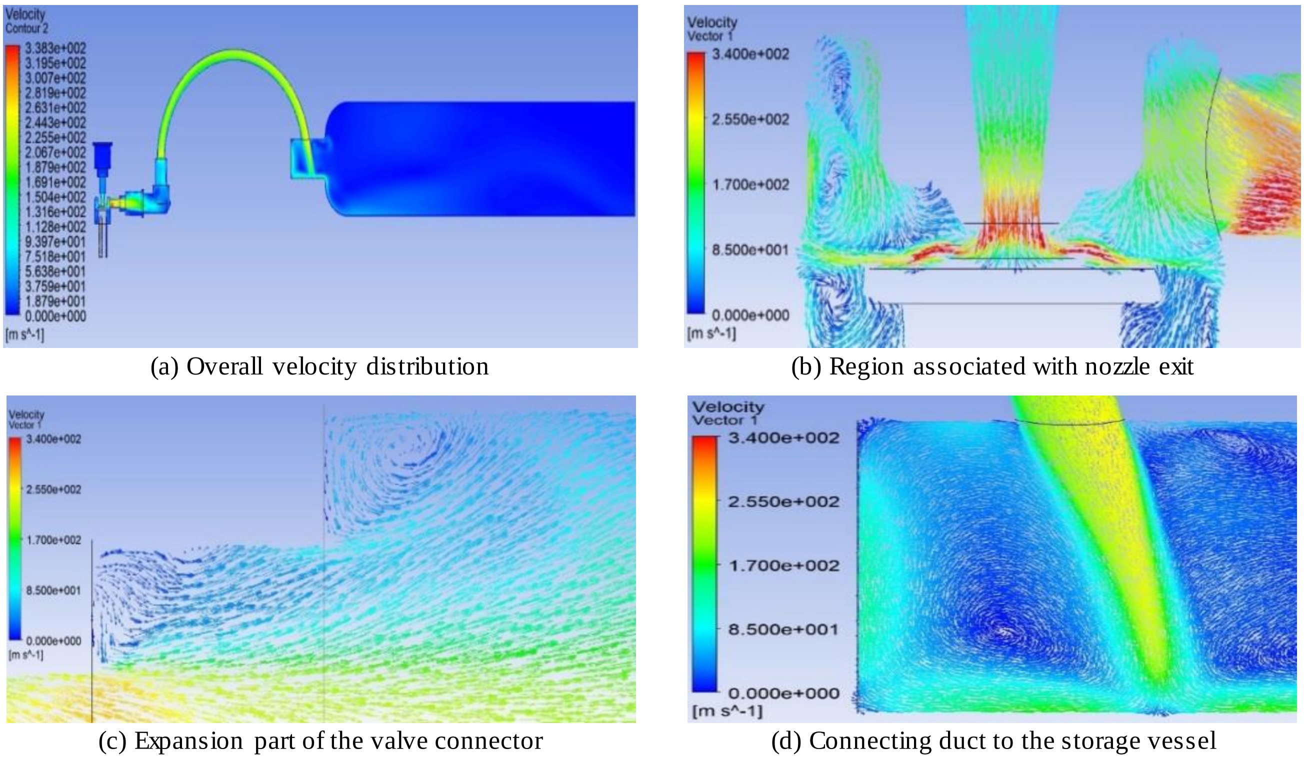
$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k\mu_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k-\rho\varepsilon \tag{4}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon\mu_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+G_{1\varepsilon}G_k\frac{\varepsilon}{k}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{5}$$

$$\mu_t=\rho C_\mu\frac{\kappa^2}{\varepsilon} \tag{6}$$

where $G_k=u_i(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i})\frac{\partial u_i}{\partial x_j}$ is the turbulent kinetic energy generated by the mean velocity gradients. $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ, σ_K, and σ_ε are model constants with the following empirically derived values: $C_{1\varepsilon}$= 1.44, $C_{2\varepsilon}$= 1.92, C_μ= 0.09, σ_K= 1.0 and σ_ε= 1.3, respectively [16].

4. Numerical Simulations and Discussions



(a) Overall velocity distribution

(b) Region associated with nozzle exit

(c) Expansion part of the valve connector

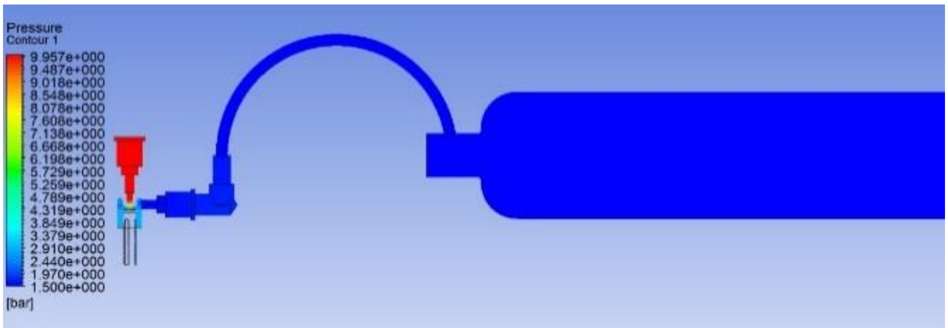
(d) Connecting duct to the storage vessel

Fig. 5 Velocity distribution for the pressure-rising process inside a pneumatic solenoid valve

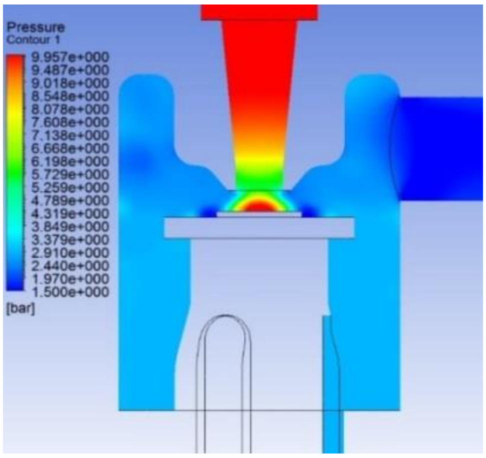
This numerical study chooses a commercial 3/2-way solenoid valve as the reference valve for analyzing its performance. Firstly, CFD software Fluent is adopted to simulate the steady flow field associated with the valve configuration. Later, with the aids of analyzing numerical results, the comprehensive flow visualization is implemented to identify the locations of adverse flow patterns, which are critical for proposing the improving alternatives. Hence, the thorough realization on performance features of this valve is attained.

Fig. 5(a) shows the overall velocity distribution inside this pneumatic solenoid valve. The high-pressure incoming air stream flows into the valve through the nozzle and undergoes an accelerating and expanding process. Then, this high-speed stream at the nozzle exit enters the small clearance between the nozzle and the armature. Certainly, the compressed air hits the armature strongly and directly before it flows into the inner space of valve. As indicated in Fig. 5(b), there are several significant circulations occurred in the right portion while a much weaker circulation exists in the right part, which is due to an air outlet provided by the connector. Later, owing to the stepwise geometry inside the connector, expansion and circulation are observed in these area-enlarging locations (see Fig. 5c). Finally, the compressed air reaches the 1-liter vessel via the connecting duct with a small cross section. Certainly, as demonstrated in Fig. 5(d), two circulations generate on both sides of the incoming air flow.

In addition, the pressure distribution in the overall system of this pneumatic solenoid valve is illustrated in Fig. 6(a). Obviously, the pressure trend decreases along the flow path from the nozzle, the connector, the connecting duct, and the storage vessel as expected. Certainly, the most dramatic pressure variation occurs inside the nozzle and region near the clearance between nozzle and armature as indicated in Fig. 6(b). As a result, the comprehensive flow visualization yields the locations of adverse flow patterns, Also, circulation and reserved flows are observed at region near the nozzle exit, expansion part of valve connector, and junction of the connecting duct to vessel. The above information is critical for proposing the improving alternatives. Accordingly, the nozzle diameter and the distance between nozzle and spool top are identified as the important design parameters to enhance the pressure response characteristics of valve.



(a) Overall system



(b) Region near the nozzle exit inside the valve

Fig. 6 Pressure distribution for the pressure-rising process inside a pneumatic solenoid valve

5. Conclusions

The flow patterns and response characteristics of a 3/2-way solenoid valve under the charging mode are analyzed in this numerical investigation. With the aids of comprehensive flow visualization, the locations of adverse flow mechanisms are realized and identified as the foundation for further modifications on its reaction performance. It follows that the locations of circulation and reserved flows are observed at region near the nozzle exit, expansion part of valve connector, and junction of the connecting duct to the storage vessel. Also, it is found that a high-pressure region exists in the region between the nozzle exit and the top of iron core. Accordingly, the nozzle diameter and the distance between nozzle and spool top are recognized as the important design parameters for improving the pressure response characteristics of solenoid valve. Clearly, the reaction time can be reduced by increasing the nozzle diameter with an appropriate distance between nozzle and spool top. Moreover, the charging time for this valve is estimated successfully via a transient CFD calculation in an acceptable deviation from test result. In conclusion, this work demonstrates a rigorous and systematic CFD scheme to evaluate the performance characteristics and to provide important information on key design parameters of the pneumatic solenoid valve

Nomenclature

C_μ、$C_{1\varepsilon}$、$C_{2\varepsilon}$	constants of standard turbulent k-ε model	ρ	fluid density
G_k	turbulent kinetic energy generated by the mean velocity gradients	ρ_a	air density
t	time	τ_{ij}	shear stress tensor
u	absolute velocity tensor	σ_k	Prandtl constant of turbulent kinetic equation
μ	turbulent viscosity	σ_ε	Prandtl constant of turbulent dissipation equation

References

[1] R. B. Van Varseveld and G. M. Bone, "Accurate position control of a pneumatic actuator using on/off solenoid valves," IEEE/ASME Transactions on Mechatronics, pp. 195-204, September 1997.

[2] A. Gentile, N. I. Giannoccaro, and G. Reina, "Experimental tests on position control of a pneumatic actuator using on/off solenoid valves," Industrial Technology, pp. 555-559, December 2002.

[3] G. Belforte, S. Mauro, and G. Mattiazzo, "A method for increasing the dynamic performance of pneumatic servo systems with digital valves," Mechatronics, vol. 14, no. 10, pp. 1105-1120, December 2004.

[4] T. A. Parlikar, W. S. Chang, Y. H. Qiu, M. D. Seeman, and D. J. Perreault, "Design and experimental implementation of an electromagnetic engine valve drive," IEEE/ASME Transactions on Mechatronics, pp. 482-494, October 2005.

[5] E. E. Topçu, İ. Yüksel, and Z. Kamış, "Development of electro-pneumatic fast switching valve and investigation of its characteristics," Mechatronics, vol. 16, no 6, pp. 365-378, July 2006.

[6] G. Tao, H. Y. Chen, Y. Y. J, and Z. B. He, "Optimal design of the magnetic field of a high-speed response solenoid valve," Journal of Materials Processing Technology, vol. 129, no. 1-3, pp. 555-558, October 2002.

[7] L. Wang, G. X. Li, C. L. Xu, X. Xi, X. J. Wu, and S. P. Sun, "Effect of characteristic parameters on the magnetic properties of solenoid valve for high-pressure common rail diesel engine," Energy Conversion and Management, vol. 127, pp. 656-666, November 2016.

[8] Z. F. Peng, C. G. Sun, R. B. Yuan, and P. Zhang, "The CFD analysis of main valve flow field and structural optimization for double-nozzle flapper servo valve," Procedia Engineering, vol. 31, pp. 115-121, 2012.

[9] Y. X. Ma and D. C. Sun, "The numerical simulation of the flow field in an electromagnetic valve," Machine Tool & Hydraulics, vol. 36, no. 1, pp. 111-113, January. 2008.

[10] Q. F. Liu, F. L. Zhao, and H. L. Bo, "Numerical simulation of the head of the direct action solenoid valve under the high temperature condition," 24th International Conf. on Nuclear Engineering, ASME Press, June 26-30, 2016.

[11] Z. Liu, X. Han, and Y. F. Liu, "Dynamic simulation of large flow solenoid valve," International Mechanical Engineering Congress and Exposition, ASME Press, November 11-17, 2016.

[12] R. Farrell, L. I. Ezekoye, and M. Rain, "Check valve flow and disk lift simulation using cfd," 2017 Pressure Vessels and Piping Conf. Paper, ASME Press, July 16-20, 2017.

[13] Ansys Fluent User's Guide-14.5, ANSYS Inc, 2012.

[14] B. E. Launder and D. B. Spalding, Lectures in mathematical & models of turbulence, London, England, July 1972.

[15] J. O. Hinze, Turbulence, McGraw-Hill Publishing Co, 1975.

[16] S. V. Patankar and D. B. Spalding, "A calculation procedure for heat mass and momentum transfer in three-dimensional parabolic flows," International Journal of Heat Mass Transfer, vol. 15, no. 10, pp. 1787-1806, October 1972.