ISSN: 2472-3703 DOI: 10.6911/WSRJ.202412_10(12).0005

Study on the Fluid Characteristics of Plug Valves Based on CFD Visualization Analysis

Xiang Feng[1], Jihan Ye[2], Biao Chen[2], Hui Li[2], Di Wang[1, *]

[1]WenZhou Polytechnic, Wenzhou, Zhejiang, 325035, China

[2]Shenjiang Valve Co., Ltd, Wenzhou, Zhejiang, 325035, China

[*] Corresponding author

Abstract

This study employs Computational Fluid Dynamics (CFD) simulation to explore the dynamic characteristics of the flow field inside plug valves under varying degrees of valve openings, aiming to propose optimized designs. A geometric model of the plug valve was developed, with appropriate boundary conditions defined. Numerical calculations were performed using the SIMPLE algorithm to ensure the precision of the simulation outcomes. The mesh configuration of the model took into account the complexities of localized flow behaviors, utilizing polyhedral mesh elements and local refinement strategies to enhance mesh quality. The experimental results indicate that the valve opening has a significant influence on the distribution of pressure and velocity fields. As the valve opening decreases, the static pressure and velocity distributions become increasingly non-uniform, particularly near the valve core, where significant pressure and velocity gradients emerge, leading to increased flow resistance. By analyzing the flow field distribution under different valve openings, this study proposes recommendations for further optimizing the plug valve design to improve flow control precision and overall operational efficiency.

Keywords

Plug valve; Flow field characteristics; Numerical simulation; Pressure field; Velocity field.

1. INTRODUCTION

The plug valve, a common fluid control device, is widely used in various industrial sectors such as petroleum, chemical, pharmaceutical, and water treatment industries. Its primary function is to regulate fluid flow by rotating the valve core, and it is favored for its simple structure, reliable sealing performance, and ease of operation. However, in complex industrial environments, plug valves must withstand harsh conditions such as high temperatures, high pressure, and corrosive media. The fluid flow within the valve body becomes increasingly complex under these circumstances, exhibiting nonlinear and unsteady hydrodynamic behavior. Optimizing the internal flow characteristics of plug valves to minimize flow resistance and energy loss, while ensuring tight sealing, has become a crucial issue in modern fluid control engineering. The goal is to improve the valve’s control accuracy and operational efficiency[1].

The performance of plug valves is influenced not only by external factors such as material and manufacturing processes but also by internal fluid flow characteristics. Flow behaviors such as pressure distribution, velocity distribution, and turbulence intensity directly determine the valve's flow control performance, flow stability, and durability. Different valve openings alter the flow channels, affecting the hydrodynamic properties of the entire system[2-4]. Reducing the valve opening not only causes local narrowing of the flow passage, increasing flow

World Scientific Research Journal Volume 10 Issue 12, 2024
ISSN: 2472-3703 DOI: 10.6911/WSRJ.202412_10(12).0005

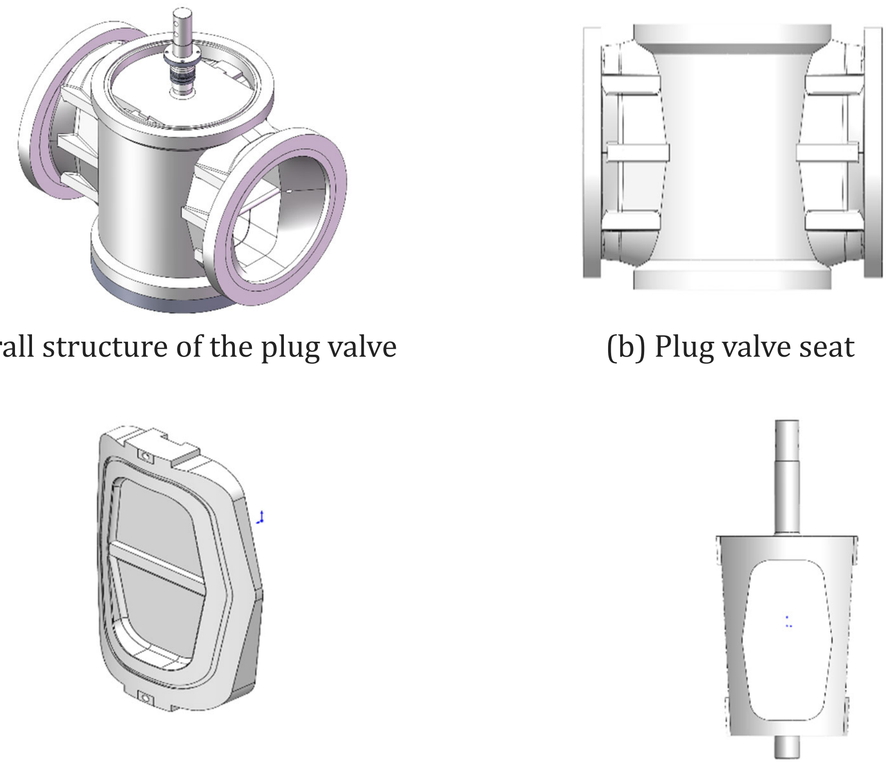
resistance, but can also intensify local turbulence, further raising kinetic energy losses. Therefore, accurately understanding the changes in internal flow fields under varying operating conditions is essential for optimizing valve design and improving operational performance[5].

Although traditional experimental methods can quantitatively analyze the flow characteristics and opening-closing performance of plug valves, these methods struggle to capture detailed information about internal flow characteristics—such as pressure and velocity distributions and vortex formation—due to the complex flow fields inside the valve body. Under high-speed and high-pressure flow conditions, the limited resolution and tolerance of experimental equipment and sensors further constrain the applicability of these methods. In response, numerical simulation technologies, particularly Computational Fluid Dynamics (CFD), have been introduced in modern engineering to offer new approaches for the design and optimization of complex fluid control systems[6-8]. In recent years, CFD has been widely applied to the hydrodynamic analysis of plug valves, particularly in exploring the impact of different valve openings on internal flow characteristics. CFD simulations allow researchers to closely observe the variations in pressure and velocity fields, assess flow resistance and energy losses under varying valve openings, and provide theoretical support for structural optimization[9].

This study employs CFD technology to investigate the dynamic fluid characteristics within plug valves, analyzing their flow field distribution patterns and evaluating their performance under different operating conditions. The findings aim to offer theoretical insights and technical guidance for the design and optimization of plug valves, facilitating their broader application in industrial settings.

2. PHYSICAL MODEL AND NUMERICAL SIMULATION METHOD

2.1. Geometric Model and Boundary Condition Setup



(a) The overall structure of the plug valve (b) Plug valve seat

(c) Plug valve disc (d) Plug valve plug

Figure 1. Plug Valve Structure Composition

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202412_10(12).0005

CFD-based computational analysis techniques provide valuable tools for in-depth investigations of the internal flow field characteristics of plug valves. Visual analysis of fluid behavior within the valve facilitates the optimization of structural design and operational parameters, minimizing potential noise, vibration, and pressure losses during actual operation. These optimizations not only improve the flow regulation precision and sealing performance of the valve but also significantly extend its service life, enhancing its overall operational efficiency and reliability. The structure of the plug valve under study is shown in Figure 1, consisting of core components such as the valve seat, valve disc, and the plug. The valve seat serves as the fixed component of the plug valve, supporting other parts and providing the fluid passage. The valve disc forms a tight fit with the valve body, acting as the primary throttling element to withstand the main fluid pressure load while ensuring sealing through surface seals. The plug, the primary moving component, moves up and down or rotates through its connection with the valve disc and stem, thus enabling the valve to open or close. By focusing on these key components, this study aims to analyze the fluid dynamics inside the plug valve and provide theoretical insights to improve its design and performance.

Boundary conditions are the constraints applied to the variables at the boundaries of the computational domain, ensuring that the solutions align with the temporal and spatial variations of the physical problem. Initial conditions define the distribution of variables at the starting point of the numerical calculations. Proper boundary conditions are essential for obtaining accurate solutions of the flow field. In transient problems, setting initial conditions is mandatory. Pressure inlet boundary conditions define the pressure at the inlet and are suitable for both compressible and incompressible flows. These are used when the inlet velocity or flow rate is unknown; in this study, the inlet pressure is set to 0.2 MPa. Pressure outlet boundary conditions specify the static pressure at the outlet boundary. This condition is only applicable when the flow velocity is subsonic. To improve convergence, pressure outlet conditions are often used to replace mass flow outlet conditions, especially when reverse flow occurs. Here, the outlet pressure is set to 0.1 MPa. Wall boundary conditions are used to restrict the fluid flow domain and separate fluid and solid regions. For viscous flows, the default no-slip boundary condition is applied, although sliding or rotating walls can be modeled to simulate specific boundary conditions. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is widely used in engineering applications for flow field calculations. It is based on pressure correction methods and staggered grid arrangements, primarily for incompressible flow problems. The algorithm operates through an iterative "guess-and-correct" approach. In this study, the SIMPLE algorithm is adopted. The fluid medium used in the calculations is listed in Table 1.

Table 1. Fluid Medium Parameters

Parameter	Value
Working fluid	Liquid water
Density	998.2kg/m3
Dynamic viscosity	0.001003kg/(m·s)

2.2. Mesh Generation and Independence Verification

To ensure accurate computational results, the plug valve model was simplified appropriately, and the inlet and outlet pipelines were extended. The inlet pipeline was extended by five times the nominal pipe diameter, and the outlet by ten times the nominal diameter, ensuring fully developed turbulent flow. The flow volume inside the plug valve was extracted for analysis, as shown in Figure 2, clearly representing the flow cross-section within the valve. The extracted

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202412_10(12).0005

fluid volume offers detailed insights into the geometric features of critical regions inside the valve.

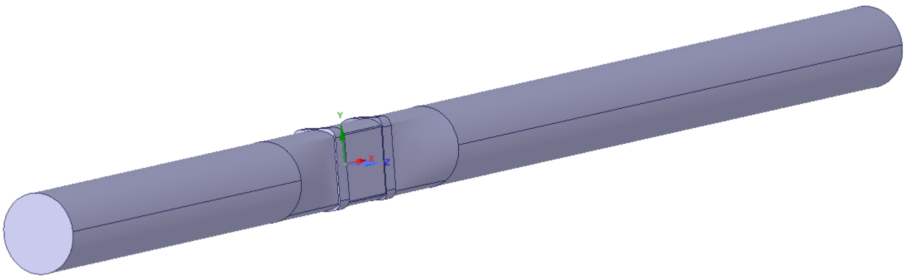


Figure 2. Internal Flow Passage of the 3D Plug Valve Model

The quality of the mesh significantly affects the accuracy of the computational results. A high-quality mesh ensures reliable simulations, especially for CFD applications. During mesh generation, special attention was given to the complex flow patterns in localized regions, particularly near the valve body and walls, where local mesh refinement was applied. The boundary layer was divided into five layers with a transition ratio of 0.2 and a growth rate of 1.2. Polyhedral cells were used for meshing due to their superior spatial filling, structural stability, adaptability, mesh quality, computational efficiency, and compatibility with complex boundaries. Figure 3 shows the generated mesh and locally refined regions. After iterative adjustments and mesh independence verification, the final mesh consisted of 111,218 cells and 367,710 nodes.

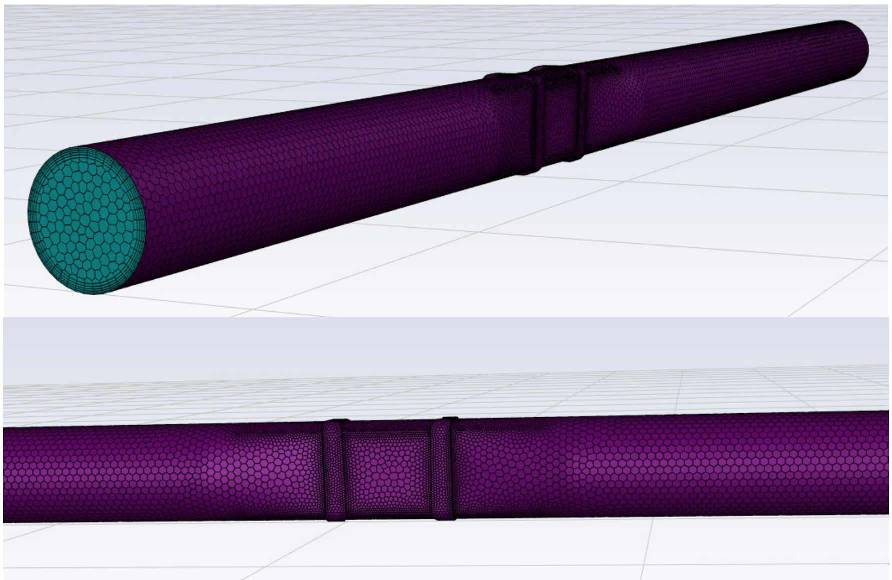


Figure 3. 3D Mesh of the Plug Valve and Local Refinement

3. EXPERIMENTAL RESULTS AND ANALYSIS

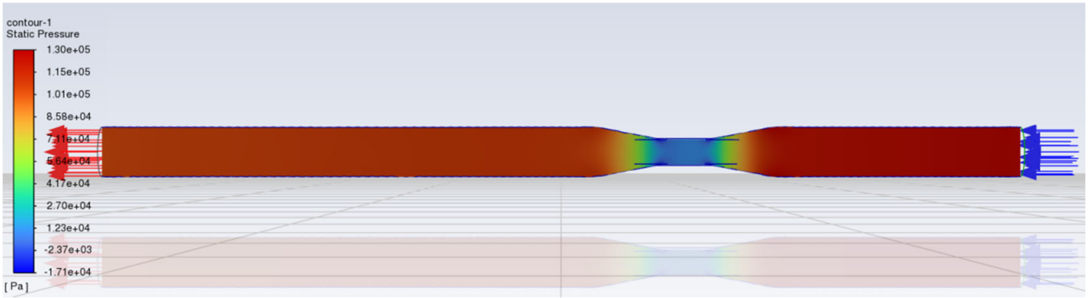
3.1. Analysis Scheme

After determining the appropriate mesh configuration, initial flow field calculations were performed. Post-processing of the results was carried out by creating different valve opening models at key positions. For a plug valve with a 100 mm pipeline diameter, numerical simulations were conducted at 20-degree intervals of plug rotation. Since fluid flow is blocked when the plug rotates to an angle below 10 degrees, states below this angle were excluded from analysis.

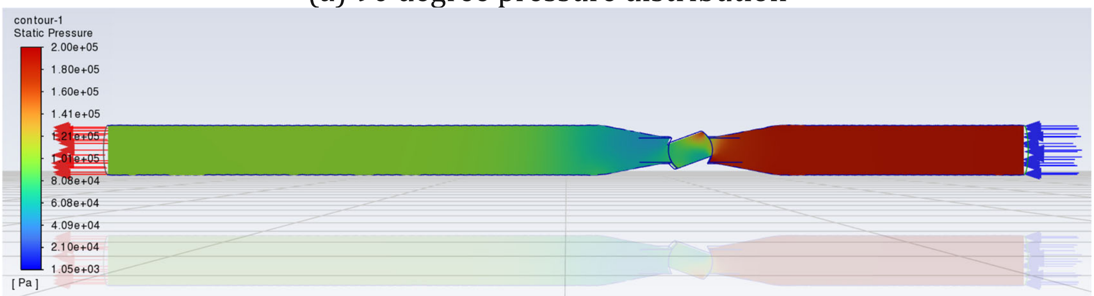
ISSN: 2472-3703 DOI: 10.6911/WSRJ.202412_10(12).0005

3.2. Pressure Field Visualization Analysis

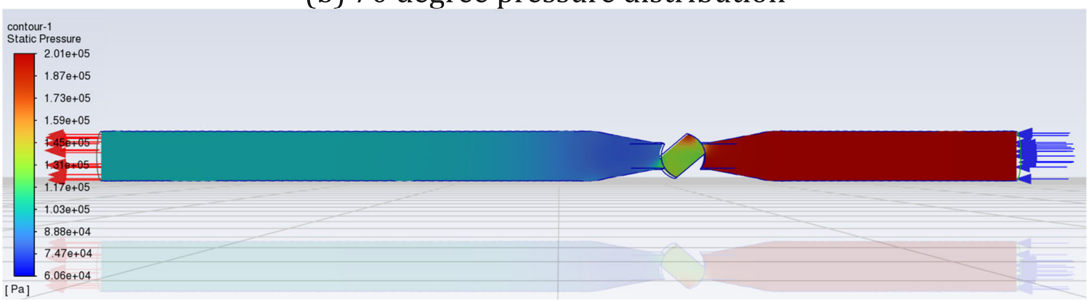
The pressure field at different valve openings was extracted from the computational results, as shown in Figure 4. At 90 degrees (fully open), the pressure distribution is relatively uniform. The inlet and outlet regions exhibit higher pressure, while the area near the valve core shows lower pressure. As fluid flows through the valve, a clear pressure gradient develops, but flow resistance remains low. At 70 degrees, the pressure distribution changes significantly. The pressure near the valve core drops sharply, and the restricted flow path increases the local pressure gradient, especially in narrow regions around the valve core. At 50 degrees, the flow path further narrows, intensifying the changes in static pressure. Strong flow blockage effects cause fluctuations in the pressure distribution, creating distinct low-pressure regions upstream and downstream of the valve core. At 30 degrees, the flow channel becomes significantly restricted, leading to asymmetrical pressure distribution. A strong pressure drop forms around the valve core, and the outlet pressure rises sharply, indicating substantial flow resistance.



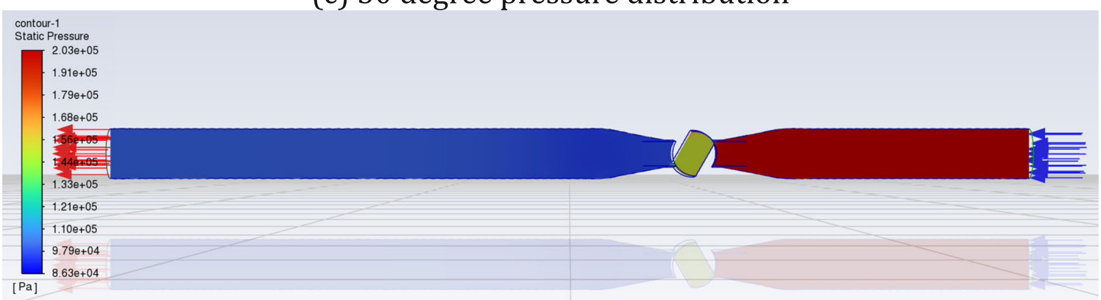
(a) 90 degree pressure distribution



(b) 70 degree pressure distribution



(c) 50 degree pressure distribution



(d) 30 degree pressure distribution

Figure 4. Pressure Distribution at Different Valve Openings

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202412_10(12).0005

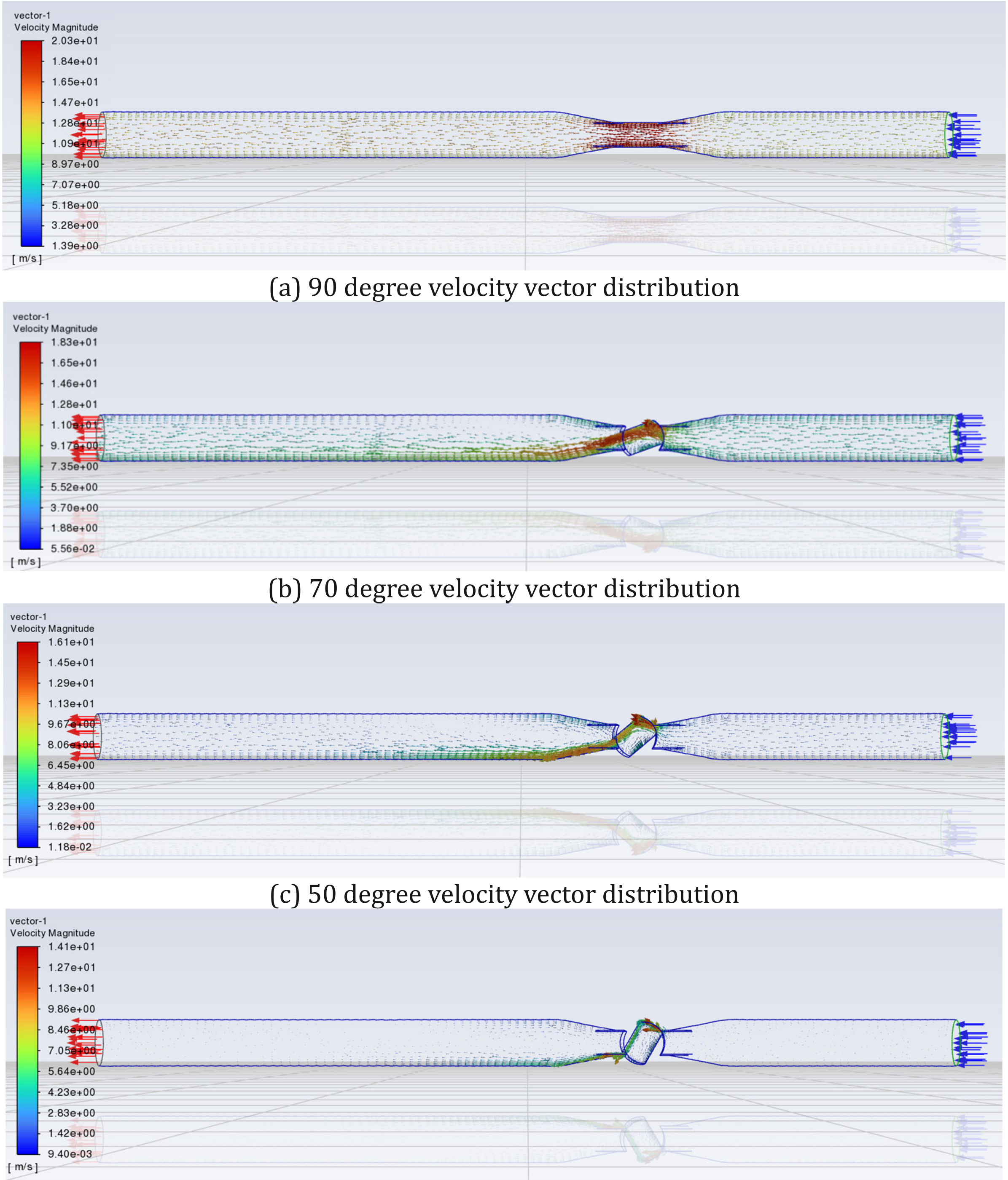
3.3. Pressure Field Visualization Analysis

Similarly, based on the computational results, the velocity vector distributions at different valve openings were obtained, as shown in Figure 5. When the plug valve is fully open at 90 degrees, the velocity distribution inside the valve remains relatively uniform. The flow rate is high and steady, especially at the inlet and outlet regions, where the fluid passes through at high speed. Due to the wide flow path in the valve core region, fluid encounters minimal resistance, resulting in smooth velocity variations. The vector distribution shows that the fluid flows orderly along the channel without significant backflow or vortex formation, indicating minimal flow resistance at this state. When the valve opening is reduced to 70 degrees, the velocity vector distribution begins to show signs of flow restriction. The flow path around the valve core narrows, leading to a notable increase in localized fluid velocity, particularly in the constricted areas of the valve core where high-velocity zones emerge. As the valve opening decreases, low-velocity zones start to develop both upstream and downstream of the valve core, with localized backflow and vortices gradually forming. These observations suggest that the fluid faces increased resistance at this point. As the opening further decreases to 50 degrees, the changes in the velocity field become more pronounced. The reduced flow area around the valve core causes a significant rise in localized velocity, with peak velocities occurring in the narrowest regions. The velocity vector distribution reveals distinct flow separation, with the backflow regions expanding and vortices becoming more prominent, especially on both sides of the valve core. This turbulent velocity distribution indicates a substantial increase in flow resistance, and the distribution of the fluid's kinetic energy becomes increasingly complex. When the valve opening is reduced to 30 degrees, the flow passage becomes severely restricted. The velocity vector distribution shows a clear separation between high-velocity and low-velocity zones. The high-speed flow intensifies around the valve core, while the low-speed regions upstream and downstream expand further. Vortex and backflow formations become increasingly prominent in symmetrical positions around the valve core, suggesting significant flow resistance and energy loss as the fluid passes through the valve.

As the valve opening decreases further, the instability of the flow rises markedly, and the extreme variations in local velocities may lead to more severe flow separation and turbulence.

As the flow field becomes increasingly turbulent at smaller valve openings, the velocity vector at a 30-degree opening was selected for local magnification, as illustrated in Figure 6. The magnified velocity vector field reveals distinct non-uniformity in the velocity distribution within the narrow regions of the plug valve. The figure demonstrates a significant increase in fluid velocity as it passes through the valve core, particularly in the high-velocity zones on both sides of the plug. In these regions, the flow direction exhibits noticeable deflections. Additionally, the vector field highlights the presence of strong velocity gradients both upstream and downstream of the valve core, which contribute to localized flow separation. In the downstream region of the valve, a clear contrast emerges between low-velocity and high-velocity zones, further indicating the complexity of the flow patterns in this area. These patterns include the formation of vortices and backflow, which provide further evidence of the intricate and unstable flow dynamics at this valve opening.

ISSN: 2472-3703 DOI: 10.6911/WSRJ.202412_10(12).0005



(a) 90 degree velocity vector distribution

(b) 70 degree velocity vector distribution

(c) 50 degree velocity vector distribution

(d) 30 degree velocity vector distribution

Figure 5. Velocity Vector Distribution at Different Valve Openings

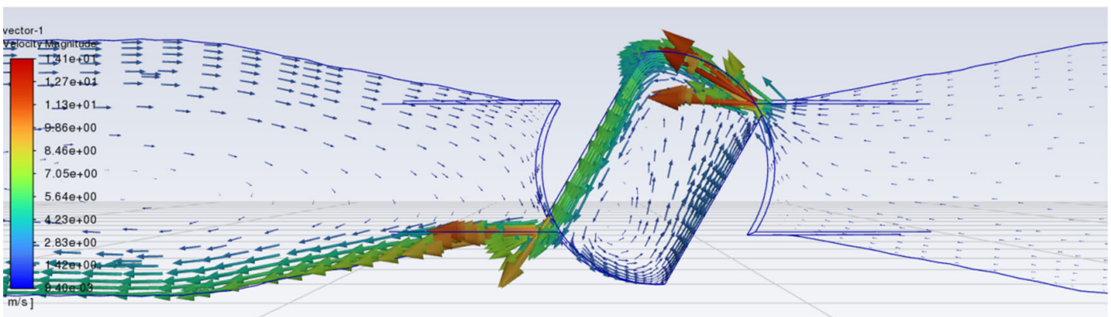


Figure 6. Enlarged View of Velocity Field at 30-Degree Opening

4. CONCLUSION

Through CFD simulations, this study systematically examined the dynamic fluid characteristics of plug valves at various opening angles. The results indicate that the valve opening significantly affects the distribution of both static and dynamic pressures. When the valve is fully open, fluid pressure and velocity distributions remain relatively uniform, resulting in minimal flow resistance. As the valve opening decreases, flow within the valve body becomes increasingly restricted, generating larger static pressure and velocity gradients. Particularly at 50 and 30 degrees, low-pressure zones and vortex formations emerge near the valve core, leading to a significant increase in flow resistance. As the valve opening continues to decrease, the flow instability becomes more pronounced, with local flow separation intensifying. These findings reveal the substantial impact of valve opening on the valve's fluid characteristics. By analyzing the pressure and velocity fields through visualization, this study provides insights into optimizing plug valve design to reduce flow resistance and improve overall system efficiency.

ACKNOWLEDGMENTS

Project: Supported by the Wenzhou Vocational and Technical College Research Program (Project Number: WZY2024031).

REFERENCES

[1] Qian J Y , Wei L , Zhang M ,et al.Flow rate analysis of compressible superheated steam through pressure reducing valves[J]. Energy, 2017, 135(sep.15):650-658.

[2] Marzio Piller, Enric Nobile,J.Thomas.DNS study of turbulent transport at low prantnumbers in a channel flow[J]. Journal Of Fluid Mechanics, 2002(458):419-441

[3] J.G.Wissink.DNS of separating low reynolds number flow in turbine casade withincoming wakes[J]. International Journal Of Heat And Fluid Flow, 2003(24):626-635.

[4] Shaw.C.T. Predicting vehicle aerodynamics Using Computational Fluid Dynamics[J]. SAESpecial Publication 747,1988:119-132.

[5] Matsunaga.K,Mijata. Finite-Difference simulation of 3D vertical flows past road vehicles[J]. Vehicle Aerodynamics,SAE Special Publication 908, 1992:65-84.

[6] Griffin.M.E,R. Diwaker,J.D.Anderson. Computational fluid dynamics applied to flows in an internal combustion engine[J]. Aerospace Science Meeting, 1978:57-78.

[7] Mampaey.F,Z.A.Xu. An experimental and simulation study of a mould filling combined with heat transfer[J]. Computational Fluid Dynamics, 1992(5):421-428.

[8] Steijsiger.C, A.M. Lankhorst, Y.R.Roman. Influence of gas phase reaction on the deposition rate of silicon carbon from the precursors Methyltrichlorosilane and hydrogen[J]. Numerical Methods in Engineering,1992(6):857-864.

[9] Bai.X.S,L.Fuchs.Numerical Model for turbulent diffusion flames with application [J]. Computational Fluid Dynamics,1992(1):169-176.