Hindawi
Journal of Engineering
Volume 2023, Article ID 2215509, 11 pages
https://doi.org/10.1155/2023/2215509

Research Article

Modal Analysis and Flow through Sectionalized Pipeline Gate Valve Using FEA and CFD Technique

Desejo Filipeson Sozinando, Bernard Xavier Tchomeni, and Alfayo Anyika Alugongo

[1]*Department of Industrial Engineering, Operations Management and Mechanical Engineering, Vaal University of Technology, Andries Potgieter Blvd, Vanderbijlpark, 1900, Private Bag X021, Vanderbijlpark 1911, South Africa*

Correspondence should be addressed to Desejo Filipeson Sozinando; desejos@vut.ac.za

Received 1 February 2023; Revised 19 April 2023; Accepted 9 May 2023; Published 25 May 2023

Academic Editor: Andras Szekrenyes

Sectionalized gate valves can reduce the volume of product released in the event of a buried pipeline failure or rupture. The risk of pipeline failure is a constant and common occurrence, and many factors can lead to pipeline incidents. In this paper, the free undamped vibration of the pipeline, sectionalized gate valve structure, and the dynamics of the fluid passing through the system are investigated. First and foremost, a modal analysis based on finite element analysis (FEA) is introduced as a fundamental linear dynamics analysis to provide insight into how a pipeline sectionalized gate valve structure may respond to different types of dynamic loading. Secondly, an implicit numerical analysis using computational fluid dynamics (CFD) is employed to describe physical quantities such as the flow velocity profiles at different stream positions and pressure fields at different points in a control volume. Through modal analysis, the effective mass factor shows the mass involved in each mode and helps identify modes with high potential to cause damage and prioritize efforts to address them. The CFD suggests that the sectionalized design of the gate valve leads to a strong vorticity of the fluid in the transversal direction of the flow and a decrease in efficiency due to pressure drop.

1. Introduction

Modern engineering practices emphasize several key factors when designing complex mechanical, aeronautical, or civil structures. Structures must maintain ever-increasing levels of lightness in order to improve efficiency and performance. Flexibility must also be incorporated to accommodate a wide variety of operational requirements. Furthermore, the design process must incorporate comprehensive studies and analyses, ensuring the structures meet the necessary standards. For example, automobile manufacturing industries are constantly investing in resources to reduce the weight of their products microscopically. It becomes progressively essential to build engineering structures by adequately considering the dynamics as we better understand them. Engineers gained a versatile conception tool using the finite element method as a computer modelling approach, particularly for examining dynamic properties. In addition to the dynamic analysis of finite element methods, this type of numerical analysis requires a rigorous theoretical orientation. The modal analysis examines the intrinsic dynamics of a system through its natural frequencies, amortization factors, and mode shapes and incorporates this information into a mathematical model to explain its behaviour. Modal model refers to the mathematical formulation of the system, and modal data refer to the information about its features. Physically, a structure's dynamics are decomposed into frequency and position. In the case of continuous systems such as beams and ropes, it is evident from the analytical solution of the equations for the partial derivatives. An invariant linear dynamic system in time can be captured through a modal analysis as a linear combination of simple harmonic movements based on the natural vibrations of the system. Yang et al. studied the process of closing and attenuating the noise of the water hammer caused by the displacement of the valve head, but not the opposite of the outflow of the fluid [1]. During this time, Lai et al. examined the mechanism of anti-check valve opening and proposed a function of approximation showing disk rotational features. In pipeline systems, relief valves are used to control maximum pressure fluctuations [2]. The CFD technique was used by Zhang et al. to analyze the purgative process and relief valve functionality and internal flow field of blow-assisted pipe [3–5]. They concluded that the geometry of the relief valve is crucial to its performance, such as the coaxially of the disk. Pipeline codes have complied with a selection of design factors, pipeline location, and maximum distance requirements between sectionalized gate valves [6]. Switching valves enable the piping systems to be turned on and off. According to [7, 8], the operation of the drive component in isolation valves for main water and feed water was investigated. In a co-simulation using Adams, UG, and MATLAB, the best pressure for motion performance was determined by developing a dimensionless objective function. Additionally, modal and structural analyses and performance assessment are important components of valve design. Vibration characteristics of the valves are assessed through modal analysis, which is achieved either numerically, experimentally, or theoretically. According to [9], the flapper nozzle valve's modal characteristics are determined numerically by superimposing its natural frequency on top of the flapper armature assembly and its squeal noise was caused by oil pressure pulsation. Zhang et al. investigated the relationship between the mass of the upper platform and the hydraulic stiffness of the leg with the electrohydraulic Stewart's sensitivity using natural frequency and mode shape analysis [10]. Lin et al. conducted CFD-DEM simulations to analyze the erosion properties of the gate valve. The simulations revealed that as the valve opening decreases, the number of particles distributed upstream of the flashboard increases [11, 12]. Modal analysis of beam structures has been performed in [13] using multiple-scale random field models. They modelled the heterogeneous material properties and their cross-correlation using random field theory. Flow resistance characteristics and internal flow characteristics of gate valves are affected by different inlet velocity in medium-low pressure gas transmission. A valve's core is often the site of pressure energy loss during gas transmission because the pressure field distribution, velocity distribution, and velocity streamline distribution depend on the valve opening degree [14].

Sectionalized gate valves enable a reduction of the volume of liquid flowing through the pipeline system. As far as the pipeline and disconnector valves are concerned, the fluid in the pipeline is the only part that is considered part of the system as a result of the pump's action. Based on the modified Bernoulli equation for incompressible fluid and the polynomial regression function, the volumetric flow rate at the operating point was determined and incorporated as an input parameter into the CFD model to predict the behaviour of the fluid passing through a sectionalized gate valve incorporated into the pipeline system.

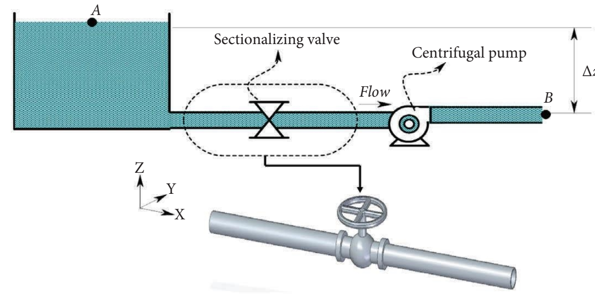


Figure 1: Schematic of an ideal model of the pipe sectionalized gate valve system.

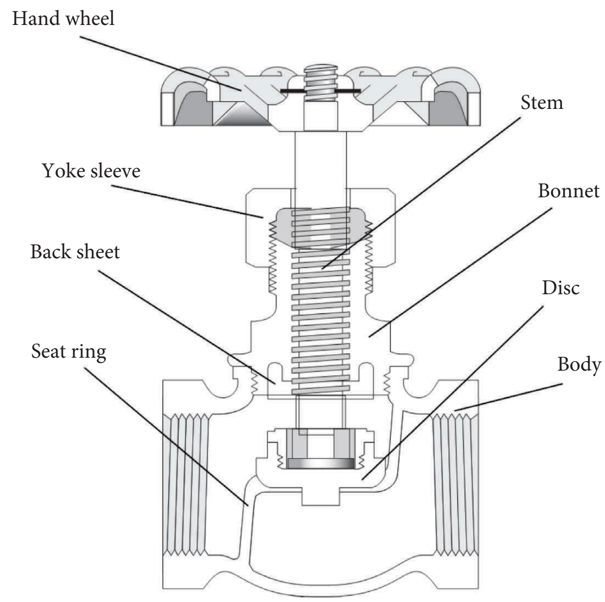


Figure 2: Two-dimensional model of gate valve.

2. Ideal Sectionalized Gate Valve Pipe System

The sectionalized gate valve pipe system model shown in Figure 1 is a visual representation of a two-dimensional pipe system that shows the components and their connectivity. It is an important tool for understanding the functionality of the system. The model typically includes the gate valve, pipes, and any other relevant components, arranged in a manner that clearly illustrates their connections and relationships. This type of simplified model is essential in demonstrating the fundamental principles of a gate valve pipe system, making it an important step in the design and implementation of these systems.

The two-dimensional model in Figure 2 shows a gate valve that can be controlled by handwheels to open or close. The fluid can flow freely through the valve when the gate or disc is open, allowing it to be elevated out of the flow path. A closed position obstructs the flow of the fluid by lowering the gate or disc. A parallel seat is located on either side of the gate or the disc to guide the disc. A gate valve can only operate in two positions: fully open and fully closed, and it does not provide throttling functionality. Fully open valves reduce energy loss and improve system flow efficiency because of the minimal pressure drop across the valve.

Figure 1 illustrates a simplified model of a pipe system that includes a reservoir open to the atmosphere, a pipeline with a sectionalized gate valve, and a centrifugal pump. Fluid flow is controlled by adjusting the valve position on the gate valve in the pipeline. A centrifugal pump increases the pressure of the fluid and transports it through the pipeline from the reservoir to the centrifugal pump. Fluid flow through the pipeline can be controlled with the sectionalized gate valve. Regulating the flow of fluid can be accomplished by closing or opening the valve. An efficient way to control and manage the flow of fluid through a pipeline is by combining a reservoir, centrifugal pump, and a sectionalized gate valve.

3. General Equation Governing the Pipeline System

Table 1 and equation (1) refer to the process of using polynomial regression to model the characteristic curve of a pump. The manufacturer has provided data points, which are used to fit a polynomial equation. The order of the polynomial can be changed to produce the best fit for the data (*n*th-order). Pump head is the dependent variable in this situation, while the volume flow rate is the independent variable. As a consequence of this procedure, the pump performance for various flow rates can be projected using an interpolated pump head function. The polynomial regression method is an eigen-curve tool for predicting data trends and patterns. However, it can be prone to overfitting if the polynomial degree is too high. Additionally, it is sensitive to outliers in the data. Therefore, it is important to check and ensure that the model is suitable for the data and it is providing accurate results.

According to least-squares, this function returns a polynomial of the nth order of the volume flow rate Q and the head H that corresponds to the best data fit. It is possible to extract the coefficients of the polynomial regression function from the output polynomial regression function using the method stated in [15].

$$H_{\text{Pump}}(Q) = a_0 + a_1 Q + a_2 Q^2 + a_3 Q^3, \quad (1)$$

where a_0, a_1, a_2, and a_3 are third-order polynomial coefficients. The coefficient number is represented by a constant a_0 without a variable multiplied by it. Polynomial terms characterize each product as denoted by $a_i Q^i$. A polynomial function reaches its maximum power at the maximum power of its variable. In mathematics, a leading coefficient is a power term with the highest power. Equation (2) represents the Bernoulli equation modified to account for the flow of incompressible fluid between a reservoir's surface and its outlet in meters. The conservation of energy underpinning Bernoulli's principle is explained in the following manner:

Table 1: Centrifugal pump characteristics (CP 250A).

Q (m^3/s)	H (m)	η (%)
0	74	0
0.002	73	35
0.003	71	50
0.005	68	65
0.007	65	75
0.008	61	85
0.01	57	89
0.012	51	72
0.013	45	64
0.015	37	20

$$\frac{P_1}{\rho g} + \frac{v_1^2}{2g} + Z_1 + h_A = \frac{P_2}{\rho g} + \frac{v_2^2}{2g} + Z_2 + \sum h_L. \quad (2)$$

Pumps transferred a total pressure head to a fluid using a static pressure head $P/\rho g$, a dynamic head $v^2/2g$, and a potential head Z above a certain reference level. By summarizing all losses Σh_L experienced by a particle, if it flows from point 1 to point 2, the particle's losses (losses ranging from minor to major) are calculated.

The Darcy–Weisbach equation (equation (3)) is a common method used to calculate head loss due to friction. It relates the head loss to the fluid properties, pipe characteristics, and the flow rate. Based on this equation, engineers can predict the head loss and estimate the pressure necessary to maintain a desired flow rate in the presence of a variety of friction factors, such as the roughness of the pipe, its diameter, and its flow velocity. According to the equation below, the frictional loss, static elevation, and pump head loss are added up to form the total frictional loss:

$$h_L(d, Q) = \sum_{i=1}^{n} \left\{ \frac{8Q_i^2}{\pi^2 g d_i^4} \left(K_i + f_i(d, Q) \frac{L_i}{d_i} \right) \right\} + \Delta Z - h_A. \quad (3)$$

A completely turbulent flow is characterized by a constant friction factor (f) despite moderate changes in flow. The friction factor is a dimensionless value that describes the resistance to flow caused by internal roughness in the pipe or duct. The coefficient of minor loss (K) is another dimensionless value that describes the head loss due to changes in the flow path. In a completely turbulent flow, the head loss due to friction is typically modelled as a proportional function of the flow rate squared, which means that the head loss is directly proportional to the square of the flow rate. This relationship is known as the law of fully developed turbulent flow and is represented by the Darcy–Weisbach equation.

Based on pipe diameter, flow velocity, and fluid viscosity, Reynolds numbers (R_e) provide flow state information in the pipeline system on an immediate basis. When the fluid laminar flow is slowly moving, viscous forces dominate, and it appears that the layers are sliding over one another. In a turbulent flow, the fluid particles move chaotically and unpredictably and the inertial forces dominate the viscous forces. It is important to note that the fully turbulent flow is valid only when the Reynolds number is greater than 4000. At this point, the pipe is considered in a turbulent state, meaning that the flow in the pipe is turbulent and the friction factor is constant and independent of the flow rate. If the Reynolds number is below 4000, the flow is considered laminar and the friction factor depends on the flow rate. In the transition zone between laminar and turbulent flow, the Reynolds number is between 2300 and 4000, and the head loss can be modelled using the power-law method.

$$R_e(d, Q) = \frac{4\rho Q}{\pi d \mu}, \quad (4)$$

where Q is denoted as the pipe volume flow rate (m^3/s), ρ is the density of water at 20°C (989.68 kg/m^3), d represents the inside pipe diameter (40 mm), and μ is the fluid absolute viscosity of 1.005×10^{-3} Ns/m^2.

$$f(d, Q) = \begin{cases} \dfrac{64}{R_e(d, Q)}, & \text{if } 0 \leq R_e(d, Q) \leq 2300, \\[2ex] \dfrac{0.25}{\left[\log\left(\varepsilon/3.7d + 5.74/R_e(d, Q)^{0.9}\right)\right]^2}, & \text{otherwise.} \end{cases} \quad (5)$$

When the pipe materials vary in roughness ε of 5.004×10^{-6} m, consider how friction losses occur as a result of the fluid moving through the pipe.

In Figure 3, the pump characteristic curve is shown against the system curve, which represents the head loss in the system. The intersection points between the two curves, known as the operating point, reflect the volume and the head at which the pump is operating. An efficiency of 87.06% can be achieved with a flow rate of 0.0095 m^3/s, a static elevation of 10 m, and a head of 58.25 m. These data can be used to analyze pump performance at different flow rates and determine the optimal pump operating point. It is also possible to calculate the pump efficiency by comparing the head increase at the operating point with the total load increase at the point on the pump curve when the pump is most efficient. By analyzing this information, flaws in pump design or operation can be found.

4. Finite Element Modal Analysis of Pipeline Sectionalized Gate Valve

The modal analysis is a useful tool for understanding the dynamic response of a structure to different types of loading, such as fluid hydrodynamic forces. In the case of a pipeline sectionalized gate valve, this analysis can help to identify potential resonance vibrations along the system and inform the design to avoid these issues. The analysis involves determining the natural frequencies and normal modes of the structure, which can be used to predict how the system will respond to different types of loading. The general linear equation governing the motion of the pipeline sectionalized gate valve system

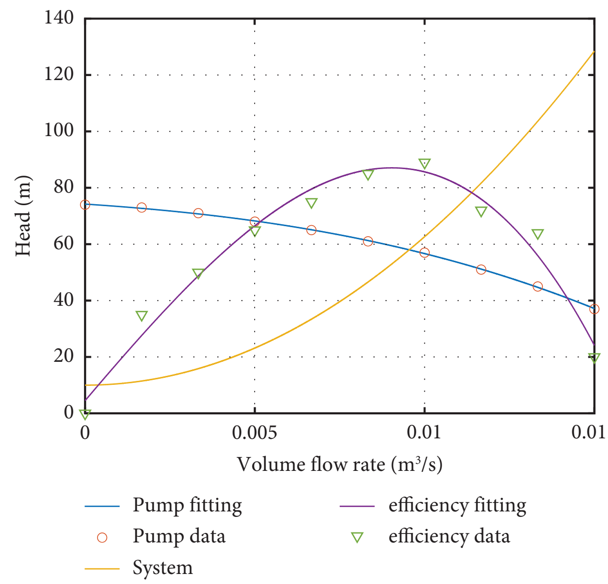


Figure 3: Pump and system characteristic curve.

Table 2: Material properties of the FEA model.

Material	Young's modulus (GPa)	Density (kg/m^3)	Poisson's ratio
Alloy steel	210	7700	0.28

can be written by ignoring the damping and external excitation and can be written as

$$[M]\{\ddot{u}\} + [K]\{u\} = \{0\}, \quad (6)$$

where M represents the mass and K represents stiffness. Assuming the displacement varies harmonically with respect to time, the displacement u and acceleration $\ddot{u}$ factors can be expressed as

$$\{u\} = \{\phi\}_i \sin(\omega_i t + \theta_i), \quad (7)$$

$$\{\ddot{u}\} = -\omega_i^2 \{\phi\}_i \sin(\omega_i t + \theta_i), \quad (8)$$

where $\{\phi\}_i$ is the modal displacement vector, and substituting (7) and (8) into (6) yields

$$\left([K] - \omega_i^2 [M]\right)\{\phi_i\} = \{0\}. \quad (9)$$

Therefore, the natural frequency f_i can be calculated as

$$f_i = \frac{\omega_i}{2\pi}. \quad (10)$$

The participation factor and effective mass give an idea of how the pipeline sectionalized gate valve structure responds to different types of dynamic loading. Participation factor measuring the amount of mass moving in X, Y, and Z directions for each mode is given as

$$\gamma_i = \{\phi\}_i^T [M]\{D\}, \quad (11)$$

Table 3: First five modes of linear vibration.

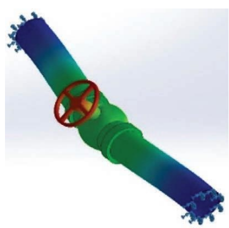
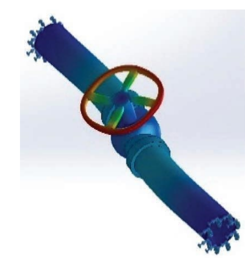
Mode shape: 1 Value = 88,8406 Hz	Mode shape: 2 Value = 97,872 Hz	Mode shape: 3 Value = 149,634 Hz	Mode shape: 4 Value = 160,331 Hz	Mode shape: 5 Value = 286,782 Hz
				
Resultant amplitude: $1.058e-01$ mm Node: 1292	Resultant amplitude: $6.840e-02$ mm Node: 1427	Resultant amplitude: $1.954e-01$ mm Node: 12218	Resultant amplitude: $1.560e-01$ mm Node: 12110	Resultant amplitude: $2.024e-01$ mm Node: 11840

Table 4: Mass participation (normalized).

Mode number	Frequency (Hertz)	X direction	Y direction	Z direction
1	88.841	$1.8274e-06$	0.71748	$2.9958e-05$
2	97.872	$1.5252e-07$	$3.4946e-05$	0.81758
3	149.63	0.14462	0.00059885	$6.7394e-07$
4	160.33	0.00062441	0.10461	$4.5468e-06$
5	286.78	$2.3448e-05$	$2.6013e-06$	$8.0217e-07$
		Sum X = 0.14527	Sum Y = 0.82273	Sum Z = 0.81761

where $\{D\}$ is the unit displacement spectrum in each of the global Cartesian directions and the rotation about each axis.

Table 2 shows the relevant parameters of material properties applied in the finite element analysis (FEA). Inside pipe diameter, the wall thickness, and total length of the model pipe section are 350 mm, 10 mm, and 3.471 m, respectively. Due to the small wall thickness to diameter and axial length of the pipe, linear elastic isotropic materials are the most effective. Finite element analysis is performed using Solidworks. The finite element simplified model of the pipeline sectionalized gate valve structure is shown in Figure 1. Grid size control and pipeline meshing are performed using automatic meshing. There are 21280 mesh nodes, 11457 elements, and a maximum aspect ratio of 209.27.

Table 3 summarizes the results of natural frequencies and the maximum displacement magnitudes for the different mode shapes using Solidworks simulation software. It is noted that the system has fixed constraints located at both ends restricting any movement. Using the maximum displacements of the sectionalized gate valve pipe structure, vibration modes relevant to the five mode shapes were identified. The colour code illustrates the deformation level, for example, the blue colour code illustrates the minimum of the deformed level displaced. Deformation levels were average for the region surrounding the green and yellow colour codes, while they were greater for the region surrounding the orange and red colour codes.

Table 4 shows the results of mass participation (normalized) obtained with finite element modal analysis. Mass participation ratios are important in determining the adequacy of deformation modes to solve dynamic problems with base motion. It is noted that the Y direction, transversal along the pipeline has about 0.82273 normalized mass participating in linear vibration modes. Many codes require that at least 80% of the mass of the system should participate in specified directions [16].

The results presented in Figure 4(a) show the displacement amplitude and corresponding frequencies of the valve structure vibration modes. This information is important in understanding the dynamic behaviour of the valve and identifying potential issues that could lead to failure. The displacement amplitude represents the magnitude of the vibration in a given direction, while the frequency represents the number of oscillations that occur in a given time. It is important to note that any external force with a frequency that matches one of the modes of valve structure vibration can cause the valve to vibrate at that frequency. This can lead to increased wear and tear on the valve and potentially cause it to fail over time. To avoid this, any external forces with frequencies that match the modes of valve structure vibration must be avoided to keep the valve vibrating under perfect conditions for a long time. The effective mass factor, as measured in Figure 4(b) for each mode, represents the amount of mass involved in that mode in a given direction of excitation. These values are expressed as a percentage of the total system mass and indicate how much of the total mass is involved in a particular mode. This information can be used to identify which mode has the most potential to cause damage to the valve and prioritize efforts to address these issues.

5. Computational Fluid Dynamics (CFD)

Fluid dynamics describe fluid motion mathematically using quantities that pertain to the fluid which are the flow pressure p, fluid viscosity ν, fluid density ρ, and flow velocity u. There are different flow velocities or flow pressures in a control volume at different points. Simulation of fluid flow is one of the most important tools used in fluid dynamics. It allows for the evaluation of variations in flow velocity and pressure at different points within a fluid domain. The Navier–Stokes equations govern the motion of the most common liquids, even though all fluids are incompressible to some extent:

$$\frac{\partial u}{\partial t} = -u\nabla u + \nu\nabla^2 u - \frac{\nabla p}{\rho} + f. \tag{12}$$

According to equation (12), the combined effects of advection $u\nabla u$ and friction can determine the acceleration $\partial u/\partial t$ of fluid, particle diffusion $\nu\nabla^2 u$, pressure gradient $\nabla p/\rho$, and body force f. Solidworks flow simulation was the tool used in CFD simulation to evaluate the flow velocity and pressure field inside the control volume. It was used to execute a simulation of fluid flow behaviour in a sectionalized gate valve pipe system detailing its dynamics.

In inlet boundary conditions, the distribution of variables needs to be specified in the flow domain at the inlet boundaries, mainly the flow velocity [17]. Table 5 shows the constraints necessary to solve the system of differential equations governing the fluid dynamics boundary value problem in which the conditions on one extreme of the interval are specified mostly where the inlet mass flow rate is known. Flow medium is water with a density of 997.56 kg/m^3 at standard sea level. The results presented in Figure 5 provide insight into the flow behaviour of a fluid as it passes through a sectionalized gate valve. The figure illustrates the flow trajectory streamlines, which show the path that the fluid takes as it flows through the valve.

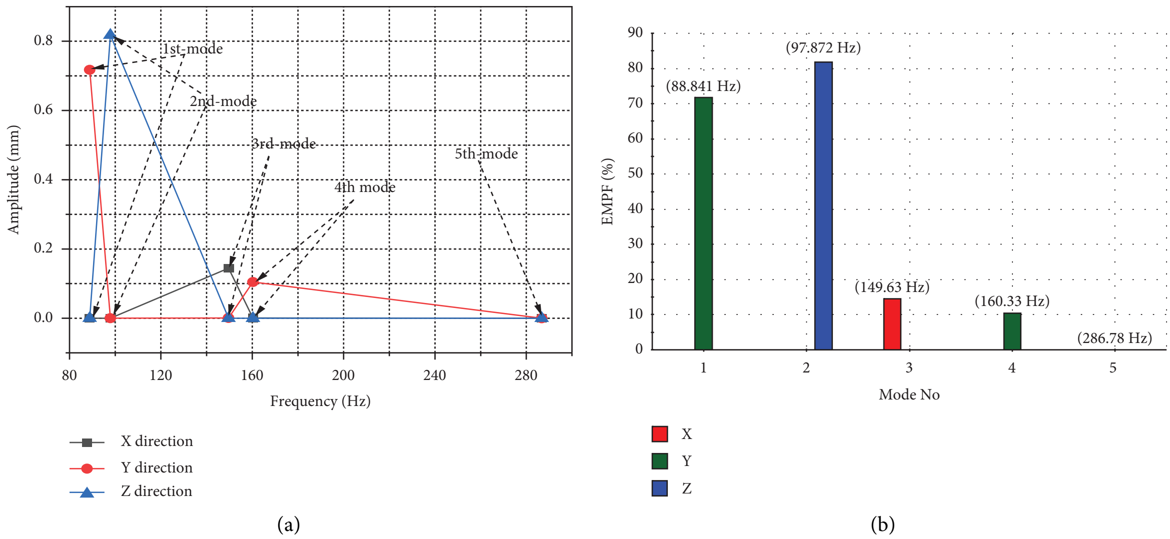


Figure 4: (a) Amplitude response of resonance frequencies. (b) Effective mass participation factor.

Table 5: Boundary conditions.

Type Reference axis	Inlet mass flow *X*
Flow parameters	Mass flow rate: 445 kg/s Fully developed flow: no
Thermodynamic parameters	Environment pressure: 101325.00 Pa Temperature: 293.20 K
Turbulence parameters	Intensity: 2.00% Length: 0.005 m
Boundary layer parameters	Boundary layer type: turbulent

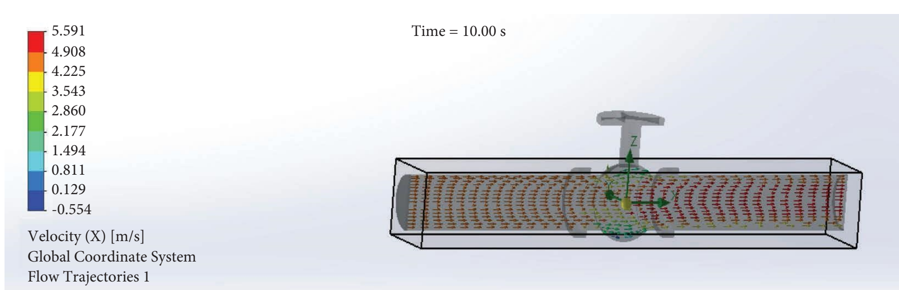


Figure 5: Fluid flow trajectories in the global coordinate control volume system.

The streamlines can be used to visualize the flow patterns and identify areas of high and low velocity, as well as regions of recirculation and vortex formation. The inlet mass flow rate of 445 kg/s is applied as the initial flow condition in the control volume in the *X* global coordinate system. This flow rate is considered a turbulent flow longitudinal to the pipe layer. The turbulent flow can be characterized by chaotic and unpredictable fluctuations in velocity, pressure, and temperature. It is usually found in systems with high Reynolds numbers, where the fluid is highly turbulent and the flow patterns are complex.

Due to the limited computation effort, the selected mesh dimensions are presented in Table 6. The selected necessary goals of physical parameters that have a substantial impact on the acceptable accuracy of the desired flow without violating the conservation of mass and momentum are shown in Table 7.

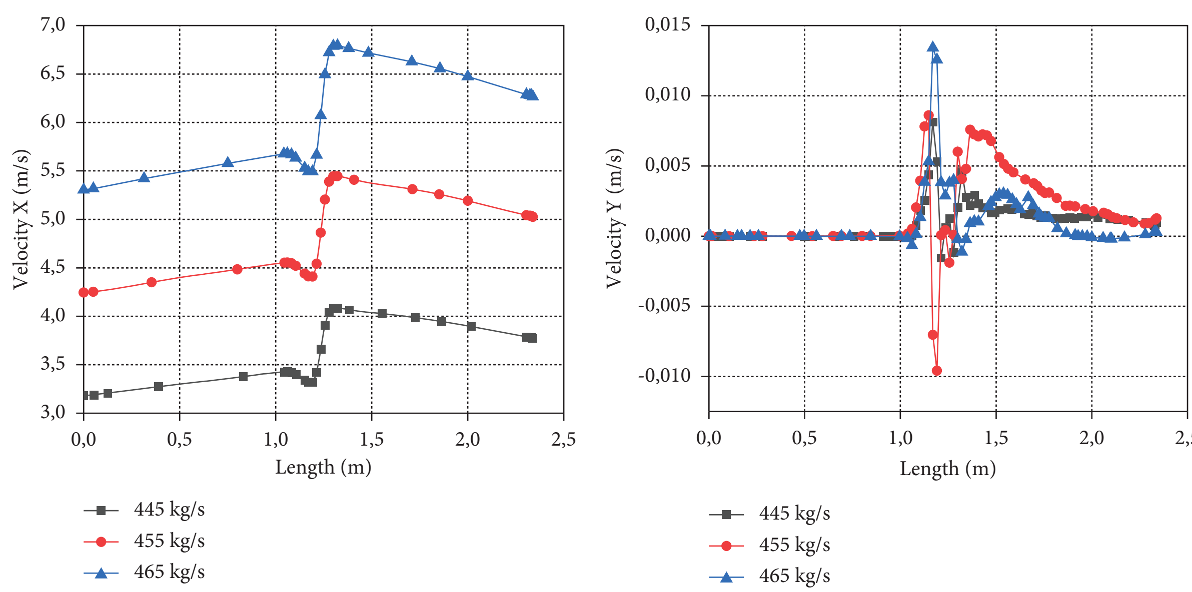
Table 6: Basic mesh dimensions.

Number of cells in *X*	36
Number of cells in *Y*	6
Number of cells in *Z*	6
Cells	4560
Fluid cells	4560

Figure 6(a) depicts the *X*-direction flow velocity profile as the fluid moves from upstream to downstream of a sectionalized gate valve. The velocity in the inlet pipe shows a small

Table 7: Goal parameters of computational fluid dynamics.

Name	Minimum	Maximum
Density (fluid) (kg/m^3)	997.56	997.56
Pressure (Pa)	97048.05	111953.23
Velocity (m/s)	0	5.591
Velocity (X) (m/s)	−0.554	5.591
Velocity (Y) (m/s)	−1.282	1.329
Velocity (Z) (m/s)	−1.195	1.309
Axial velocity (m/s)	−1.195	1.309
Relative pressure (Pa)	−4276.95	10628.23
Circumferential velocity (m/s)	−3.648	3.641
Shear stress (Pa)	0	57.94
Boundary layer thickness (m)	$3.211e-04$	0.039
Boundary layer thickness (thermal) (m)	$2.281e-04$	0.039
Boundary layer type	0	1.0000000
Acoustic power (W/m^3)	0	$7.915e-12$
Acoustic power level (dB)	0	8.98

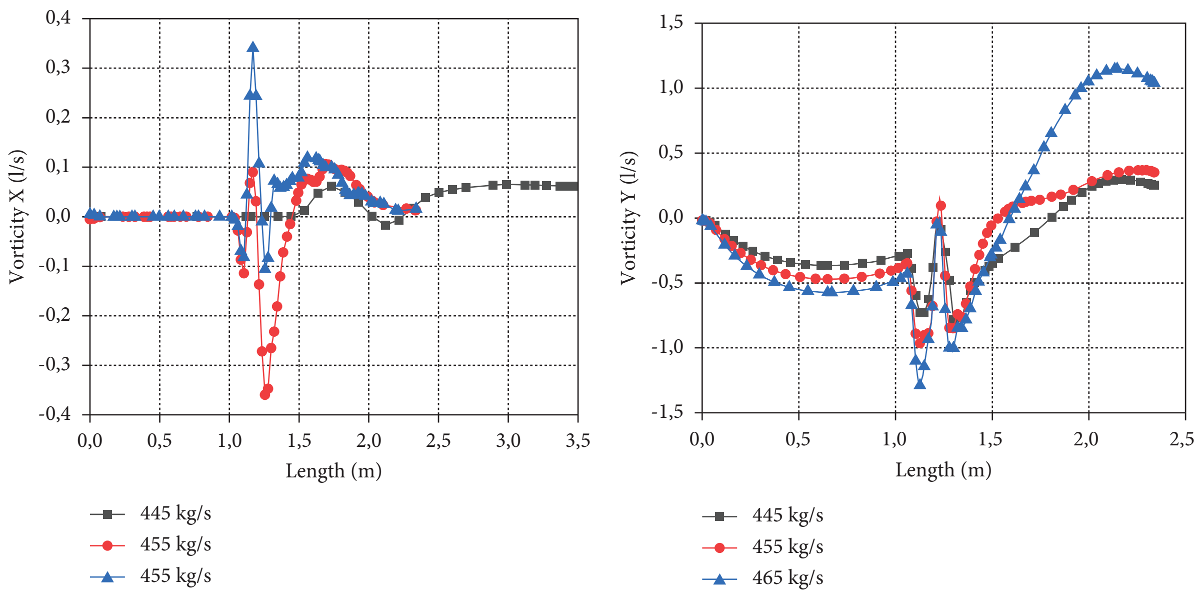


(a) (b)

Figure 6: Flow velocity field along pipe length: (a) X direction; (b) Y direction.

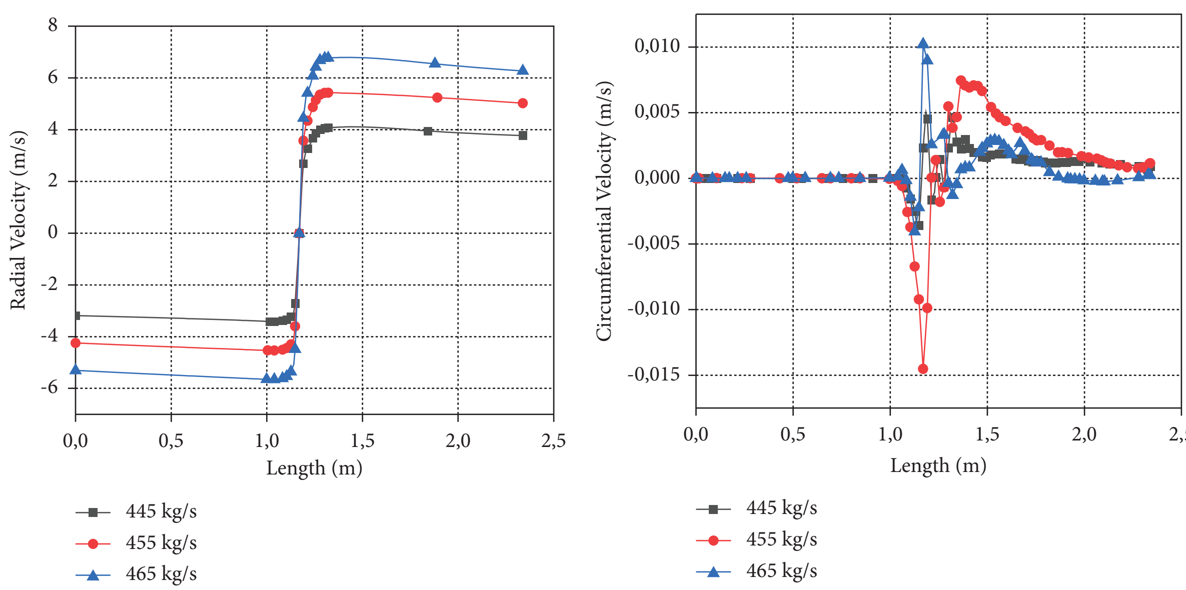
linear increase, but at the sectionalized gate valve, there is an instantaneous fluctuation due to the enlargement of the internal valve area. However, downstream of the valve, there is an exponential increase in velocity which could be caused by the local rotation of the fluid, known as vorticity, between the valve and pipe. It is clearly noted that the amount of mass that passes through the valve is proportional to the flow velocity along the streamline. In contrast, Figure 6(b) shows the Y-direction flow velocity field, which disappears at the inlet section of the valve. As the fluid passes through the valve, there is a significant fluctuation in the nonlinear behaviour of the flow velocity in the Y direction, with small variations downstream of the sectionalized gate valve.

The results shown in Figures 7(a) and 7(b) provide insight into the rotation of the fluid flow through a sectionalized pipeline gate valve. As stated in the figure's caption, the vorticity along the X and Y directions of the flow is being measured. The result in Figure 7(a) indicates that in the longitudinal flow direction (X), vorticity is generated by the change in flow direction caused by the valve's sectionalized design. This means that the design of the valve causes the fluid to rotate or swirl. A higher impact of vorticity in the Y direction of the flow, as shown in Figure 7(b), suggests that there is a stronger rotation or swirling of the fluid in this direction compared to other directions. This can be caused by various factors such as the presence of vortices, turbulence, or shear in the flow. These factors can alter the flow patterns and cause pressure drop across the valve, which can impact the efficiency of the pipeline system. It is worth noting that the vorticity values in Figures 7(a) and 7(b) are relative to the system and may not have an absolute value that can be compared with other systems. Therefore, the results should be interpreted in the context of the specific pipeline system being studied.



(a) (b)

Figure 7: Flow vorticity along pipe length: (a) X direction; (b) Y direction.



(a) (b)

Figure 8: Fluid velocity profile in a gate valve section: (a) radial velocity; (b) circumferential velocity.

Figure 8(a) specifically shows the radial velocity of the fluid, which is a measure of how fast the fluid is moving in a radial direction relative to the valve. The graph shows that there is a symmetrical instantaneous change when the flow reaches the valve centerline. This means that at the centerline, the fluid's radial velocity abruptly changes from a negative value to a positive value. Figure 8(b) shows the circumferential velocity of the fluid, which is a measure of how fast the fluid is moving in a circular direction around the valve. Throughout the internal section of the gate valve, the map shows the distribution of circumferential velocity. It is also discussed in a comprehensive and generalized way how vorticity is transferred as a measure of fluid rotation. Furthermore, the study observes a transient behaviour in the circumferential velocity when the fluid flow reaches the valve. Looking at the inlet section side of the valve, it can be seen that the circumferential velocity is zero until the flow reaches the valve. The velocity of the fluid changes as it passes through the valve as a result of transient behaviour. In addition, the study notes that attenuation at the outlet section of the valve results in a decrease in fluid velocity as it leaves the valve.

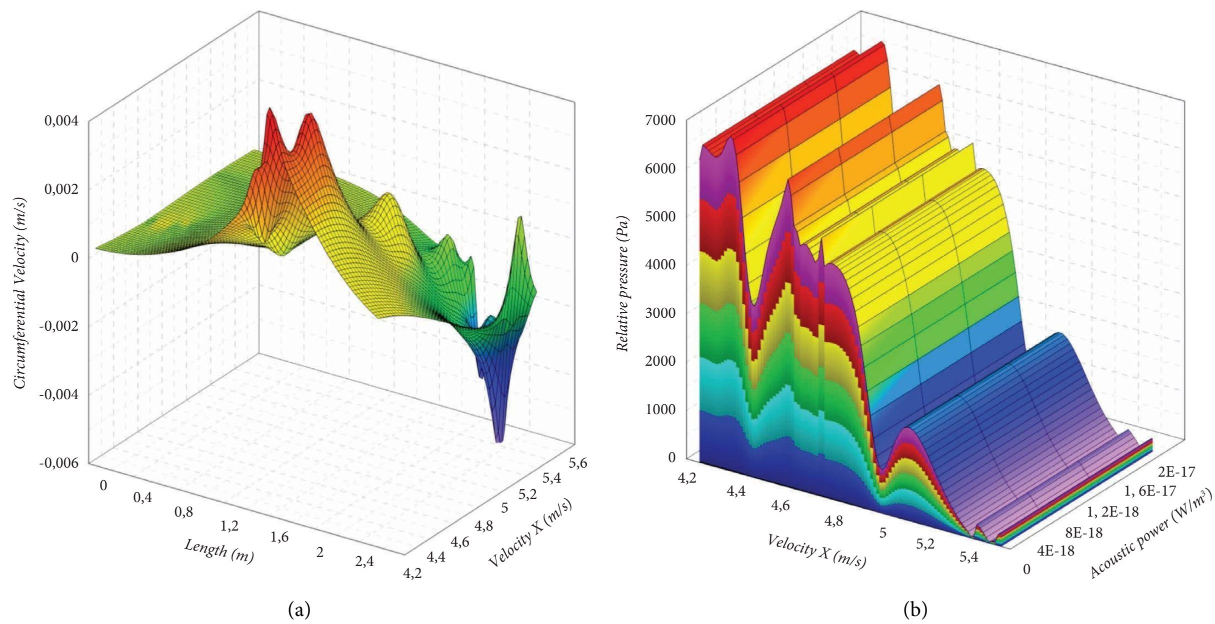


Figure 9: (a) 3D circumferential velocity. (b) Relative pressure fluctuation.

Figure 9(a) displays three-dimensional (3D) circumferential velocity and longitudinal velocity over a pipe length. A fluid flows circumferentially around the pipe, while a fluid flows longitudinally along the pipe. During a circumferential flow, two different speed components are measured: circumferential and longitudinal velocity. According to the figure, the circumferential velocity at the valve has a lower influence than the longitudinal velocity. It is observed, however, that both velocities exhibit transient behaviour near the limit of the valve. Accordingly, the longitudinal and circumferential velocities of the fluid change abruptly as it passes through the valve. According to Figure 9(b), the relative pressure fluctuations of the pipe stream are shown in the 3D map. A pressure fluctuation measurement is used to measure how the fluid pressure changes as it flows through a pipe. In comparison to the rest of the pipe, the valve shows a higher and more nonlinear pressure fluctuation. There is a sudden change in flow direction and velocity at the valve that can cause this phenomenon. Moreover, it noted that the pressure fluctuation decreases as the fluid passes through the sectionalized gate valve and exits, indicating considerable attenuation downstream.

6. Conclusion

The combination of mathematical techniques and numerical analysis was used to study the performance and behaviour of a sectionalized gate valve pipeline system. Specifically, we used Bernoulli's principle and a third-order polynomial regression with a linear iteration algorithm to determine the characteristics of the centrifugal pump system at its operating point. The system was solved numerically in a systematic way, taking into account even minor losses in addition to friction losses. Furthermore, the study used FEA modal analysis to determine how the system deforms under external forces. To prevent increased wear and failure, the study emphasizes the importance of avoiding external forces with matching frequencies. For each mode, the effective mass factor shows how much total system mass goes into each mode, and it can be used to identify the modes that pose the biggest risk to the valve. Furthermore, CFD techniques were used in order to analyze the fluid flow through a sectionalized gate valve. It was found that the design of the valve caused the fluid to rotate and swirl, with a stronger rotation in the transversal direction of the flow. This can result in a pressure drop across the valve and impact the efficiency of the pipeline system. It was also observed that both the circumferential and longitudinal velocities have transient behaviour at the limit of the valve, and the pressure fluctuation is higher and more nonlinear at the valve compared to other parts of the pipe. Some recommendations for future research include the use of other mathematical and numerical techniques to analyze the system's performance and behaviour, such as neural networks, machine learning, and genetic algorithms. The study of the effect of different types of fluids and their properties on the overall system can also be performed.

Data Availability

Data used for the findings of the study are available on request from the corresponding author.

Conflicts of Interest

The authors declare that they have no conflicts of interest.

Acknowledgments

The authors acknowledge the Industrial Engineering, Operations Management, and Mechanical Engineering departments at the Vaal University of Technology for their technical assistance in accomplishing the research objectives. Open Access funding enabled and organized by SANLiC Gold.

References

[1] Z. Yang, L. Zhou, H. Dou, C. Lu, and X. Luan, "Water hammer analysis when switching of parallel pumps based on contra-motion check valve," *Annals of Nuclear Energy*, vol. 139, Article ID 107275, 2020.

[2] Z. Lai, B. Karney, S. Yang, D. Wu, and F. Zhang, "Transient performance of a dual disc check valve during the opening period," *Annals of Nuclear Energy*, vol. 101, pp. 15–22, 2017.

[3] Z. Zhang, L. Jia, and L. Yang, "Numerical simulation study on the opening process of the atmospheric relief valve," *Nuclear Engineering and Design*, vol. 351, pp. 106–115, 2019.

[4] L. Yang, Z. Wang, W. Dempster, X. Yu, and S. T. Tu, "Experiments and transient simulation on spring-loaded pressure relief valve under high temperature and high-pressure steam conditions," *Journal of Loss Prevention in the Process Industries*, vol. 45, pp. 133–146, 2017.

[5] B. Miao and F. Chen, "Optimization simulation of ultrafine dry powder blow-assisted pipe based on CFD," *Mathematical Problems in Engineering*, vol. 2022, Article ID 8304007, 19 pages, 2022.

[6] N. K. Nath, I. Al Awadhi, and S. Narayanan, "Best practices and challenges for pipeline design and construction in populated area," in *Proceedings of the InAbu Dhabi International Petroleum Exhibition & Conference. OnePetro*, Abu Dhabi, UAE, November 2016.

[7] D. Pan, S. Gu, G. Guo, H. Kuang, H. Zhong, and F. Gao, "Co-simulation design and experimental study on the hydraulic–pneumatic-powered driving system of main steam and feed water isolation valves for CAP1400," *Advances in Mechanical Engineering*, vol. 9, no. 8, Article ID 168781401772007, 2017.

[8] D. Pan, S. Gu, G. Guo, H. Kuang, H. Zhong, and F. Gao, "The optimization methodology research on the motion performances of the driving device of main steam and feed water isolation valves," *Proceedings of the Institution of Mechanical Engineers - Part C: Journal of Mechanical Engineering Science*, vol. 232, no. 6, pp. 1069–1078, 2018.

[9] M. Chen, D. Xiang, S. Li, and C. Zou, "Suppression of squeal noise excited by the pressure pulsation from the flapper-nozzle valve inside a hydraulic energy system," *Energies*, vol. 11, no. 4, p. 955, 2018.

[10] L. Zhang, X. Zong, Y. Tang, X. Chen, J. Feng, and X. Yuan, "Modal and natural frequency sensitivity analysis of electrohydraulic Stewart platform," *Shock and Vibration*, vol. 2021, Article ID 5587282, pp. 1–18, 2021.

[11] Z. Lin, X. Sun, T. Yu, Y. Zhang, Y. Li, and Z. Zhu, "Gas–solid two-phase flow and erosion calculation of gate valve based on the CFD-DEM model," *Powder Technology*, vol. 366, pp. 395–407, 2020.

[12] E. Žic, P. Banko, and L. Lešnik, "Hydraulic analysis of gate valve using computational fluid dynamics (CFD)," *Scientific Review Engineering and Environmental Studies (SREES)*, vol. 29, no. 3, pp. 275–288, 2020.

[13] D. Feng, "Modal analysis of beam structures with random field models at multiple scales," *Advances in Civil Engineering*, vol. 2021, pp. 1–15, 2021 Jan 4.

[14] Z. Lin, G. Ma, B. Cui, Y. Li, Z. Zhu, and N. Tong, "Influence of flashboard location on flow resistance properties and internal features of gate valve under the variable condition," *Journal of Natural Gas Science and Engineering*, vol. 33, pp. 108–117, 2016.

[15] D. F. Sozinando, V. Netshitandani, X. T. Kouejou, and A. A. Alugongo, "Flow balance and pressure drop of pumping station system during routine maintenance activities," in *Proceedings of the In2022 IEEE 13th International Conference on Mechanical and Intelligent Manufacturing Technologies*, pp. 230–235, Cape Town, South Africa, May 2022.

[16] 2022,https://help.solidworks.com/2021/english/SolidWorks/cworks/HIDD_SIM_HELP_Mass_Participation.htm.

[17] C. García-Sánchez, D. A. Philips, and C. Gorlé, "Quantifying inflow uncertainties for CFD simulations of the flow in downtown Oklahoma City," *Building and Environment*, vol. 78, pp. 118–129, 2014.