

## Research Article

# Analysis of Cylindrical Damper Effects on Turbine Meters Accuracy in a Pulsating CNG Suction Line: An Optimal Design through CFD Simulations

Mostafa Faramarzi,<sup>1</sup> Saeed Dinarvand ,<sup>2</sup> Mahmood Farzaneh-Gord ,<sup>3</sup> and Hossein Tamim <sup>4</sup>

<sup>1</sup>Department of Mechanical Engineering, Shahrood University of Technology, Shahrood, Iran

<sup>2</sup>Department of Mechanical Engineering, Central Tehran Branch, Islamic Azad University, Tehran, Iran

<sup>3</sup>Department of Mechanical Engineering, Ferdowsi University of Mashhad, Mashhad 91775-111, Iran

<sup>4</sup>Department of Mechanical Engineering, Arak Branch, Islamic Azad University, Arak, Iran

Correspondence should be addressed to Saeed Dinarvand; saeed\_dinarvand@yahoo.com

Received 22 September 2021; Revised 16 February 2022; Accepted 28 March 2022; Published 18 April 2022

Academic Editor: Junwu Wang

Copyright © 2022 Mostafa Faramarzi et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

Turbine meters are used in compressed natural gas (CNG) stations in abundance to measure the volumetric flow rate of the inlet gas. These types of meters are very sensitive to oscillating and pulsating flows. When the station's reciprocating compressor starts to work, due to the piston and valve performance, pulsating flow will create in the suction line and downstream of turbine meter. This pulsation makes false pulses in the meter and finally makes difference between the measured gas flow from the metering of the gas company and the amount of gas that has been sold at the station. In this study, numerical simulation of dampeners has been investigated after turbine meter and before entering gas to compressor. The goal is eliminating or reducing the effects of operating compressor in CNG station on turbine meter and reducing measurement errors. Numerical simulation of dampener was investigated by Ansys-Fluent CFD software for a CNG station with the gas inlet pressure 250 psi, inlet and outlet pipe diameter of 2 inches. In this study, several parameters such as (1) height-to-diameter ratio in the cylinder, (2) distance between dampener and compressor, and (3) volume-to-minimum volume ratio according to API 618, have been checked. The numerical results show that by increasing height-to-diameter ratio up to 3, the pulsating amplitude on Plane 1 will be reduced, and this is the best ratio according to the present results. Also, by increasing the outlet pipe length, the pressure pulsation amplitude decreases from 3.4% of pressure line in 2 m length to 0.7% in 5 m length. Moreover, the results showed 2.4% increase and 1.9% decrease in the maximum and minimum pressure for  $CV = 1$  and  $CV = 16.7$ , respectively. Comparing fluent numerical method with previous studies shows less than 2% difference that demonstrates the validity and reliability of present investigation.

## 1. Introduction

Lower cost and lower destructive environmental effects in comparison to gasoil and gasoline caused natural gas, in different forms, to be used as one of the common automobile fuel types in many points of the world for the last few decades. One of its most widely used types is compressed natural gas (CNG) which is mainly formed by methane [1, 2]. This increasing trend in the use of CNG attracted much attention of experts and scientists working in this area

to do experimental, numerical, and analytical research studies.

Naturally, compressors are employed for producing CNG. The compressors commonly used in CNG stations are positive displacement and reciprocating models. In positive displacement compressors, pulsations may be created in charging and discharging lines due to the gates opening and closing movements [3, 4]. Clearly, these pulsations affect the accuracy of the measurement. A comprehensive study has been investigated by Farzaneh-Gordi et al. [5] to decrease the

compressor effects on measurement accuracy utilizing a snubber between the turbine flow meter and the reciprocating compressors. Detailed information about compressors pulsations and controlling methods and especially their expansion properties could be found in literature [6–8]. In the last years, much research has been conducted for finding novel and efficient methods of reducing CNG compressors pulsations and omitting their flow pulsations. In this context, Majic et al. [9] investigated the effects of discharge section port shape of screw compressors on their pulsations. Presenting a mathematical model, they showed that discharge track surface function is an important and effective parameter in creating pulsation flow and modifying this area pulsation flow could be reduced. Wu et al. [10] simulated discharge pressure pulsations in twin screw compressors and solved one-dimensional equation of unsteady-state gas flow which presents the discharge pressure pulsations in compressors by employing Lax-Wendorff two-stage numerical solution method. Qian et al. [11] simulated the pressure pulsations of Francis turbines with air suction numerically and analytically. They, actually, presented a three-dimensional simulation of the unsteady multiphase flow through a hydraulic Francis turbine and showed that suction air reduces pressure difference in horizontal section of draft conduit.

Experimental and theoretical research has been performed to determine the pulsation effects on turbine flow meters' measurement accuracy. Atkinson [12] solved the turbine rotor motion equation and employed the magnetic pickup registering the passing of a rotor blade to compute the real volume flow. This approach should only be employed if the amplitude of the pulsations is detectable by the turbine signal. By the rapid reduction of the amplitude of pulsation in the turbine meter signal due to an increase in frequency, it was difficult to forecast the real flow in the cases of pulsations with high frequency. Lee, Cheesewright, and Clark [13] developed another software tool called the "Watchdog System". A comprehensive transient numerical calculations using CFD are carried out under different number of impeller blades for the flow field within a centrifugal pump under single-phase and cavitation condition [14, 15]. Vetter and Schweinfurther [16] assessed pressure pulsations in piping section of reciprocating pumps and presented accurate patterns for various installation configurations of this type of pump employing a calculation method and comparing their experimental and computational data. Notzen [17] then tried to find applicable pulsation reduction methods for high-pressure reciprocating pumps. He proposed various methods for this objective and analyzing the pulsations numerically showed how to control and optimize the decreasing effects.

Jeong et al. [18] accomplished a numerical analysis on different models of pressure pulsation dampers at hydrogen compressors outlet. They compared pressure pulsations and pressure drop through pulsation dampener in laminar and turbulent flows by employing Star CD code and demonstrated that the best pulsation dampener is the one which presents the lowest pressure drop and pressure pulsations. Al-Obiadi investigated the effects of different turbulence

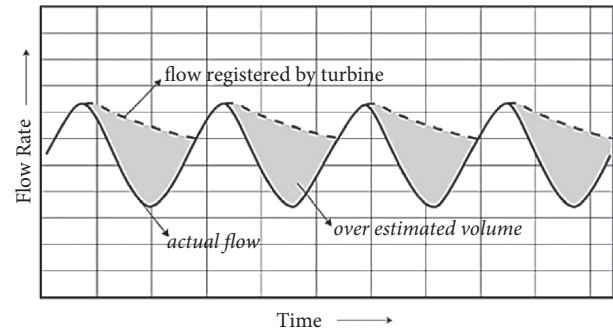


FIGURE 1: Comparison of flow registered by turbine and actual flow during time due to compressor [30].

models on three-dimensional unsteady cavitating flows and pressure fluctuations within an axial flow pump under transient flow pattern [19–21]. The numerical results reveal that the SST turbulence model was closer to the experimental results in predicting head. In addition, the pressure variation trend for the 10 models is very similar which increases and then decreases from the inlet to outlet of the pump along the streamline. The SST  $k-\omega$  model predicts that the performance of the pump was more accurate than other turbulent models. Furthermore, the results also found that the error is the least at design operation condition 300 (L/min), which is around 1.98% for the SST model and 2.14% and 2.38% for the LES and transition omega model. The use of various techniques such as vibration and acoustic analyses can provide a more robust detection of cavitation within the pump. The flow structure within a pump is very complex due to the presence of a rotating impeller and its interaction with the volute casing and mostly due to the occurrence of cavitation. Several researchers carried out an experimental investigation using acoustic analysis technique to determine the pump performance and detection of cavitation within a centrifugal pump [22, 23].

The most important component of a CNG station is a reciprocating compressor. In a CNG station, the pressure of the gas must be increased from approximately 0.5 MPa to about 20 to 25 MPa. A multistage compressor is employed to achieve this pressure increment. The largest share of the current and initial costs of the stations depends on the reciprocating compressor [24]. Turbine gas meters (TGMs) are employed plenty for measuring volumetric flow rate of inlet gas in the CNG stations [25, 26]. The rotational speed of the turbine wheel is proportional to the mean gas velocity and, therefore, to the gas flow rate [27]. The number of rotations is a measure of the gas volume that has flown through. These types of meters are inferential meters and therefore dependent on the direction, distribution, and homogeneity of the gas flow, and they must be used in steady flows [28]. Turbine meters have acceptable accuracy in engineering applications, but because of the inertia of the rotor, they are effective to the pulsation flows. You can find pulsation flows in charging line of positive displacement compressors that commonly used in CNG stations. Sun et al. [29] studied pressure flows in suction chamber and

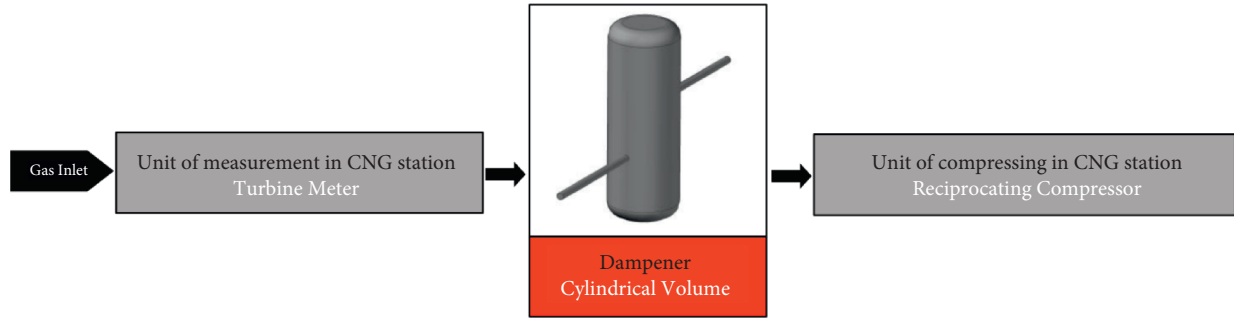


FIGURE 2: Schematic of flow direction and dampener position between turbine meter and compressor.

presented that in compressors, pulsation amplitude is about 5% of line pressure and pulsation were nearly sinusoidal.

The turbine wheel slows down after the flow has stopped by means of a valve, and the measured gas volume is always higher than that which has actually flown through the gas meter. In pulsation gas flow, the error of gas turbine is positive because the extra-rotation of the rotor produces an overestimated gas volume. Figure 1 shows that flow registered by turbine is always more than actual flow [30]. This error makes difference between the amount of gas that have registered by Gas Company and sold by CNG station. It should also be noted that there are two alternative CNG measurement methods which Cascetta et al. [31], carried out a comparison between these flow measurement approaches (volumetric versus mass flow metering), and 10% deviation was observed.

Standard API 618 [32] covers the minimum requirements for reciprocating compressors in oil, gas, and petrochemical industries. One of the most important issues addressed in this standard is pressure pulsation made in such compressors which may cause sharp pulsations and serious damages to the compressor suction part belongings as well as causing limitation for higher pressures [33]. In addition, according to former paragraph, such pulsations at the suction part can make considerable uncertainties in measurement systems of the CNG station. Therefore, according to API 618, installing appropriate pulsation dampeners at reciprocating compressors inlet for high pressures is vital. Figure 2 shows a schematic of the dampener position between the compressor and turbine meter and illustrates the direction of flow. Although API 618 clearly proposes the lowest required volume and minimum allowable pressure pulsation compared to pressure line for the dampener, it does not give any information about the optimal value of height-to-diameter ratio ( $H/D$ ) of the device and for finding optimal volume for dampener, the lowest required volume and minimum allowable pressure pulsation must be investigated. This study, therefore, aims to find the optimal value of  $H/D$  and pulsation dampener volume which the lowest rate of pressure pulsations, at a specific and constant pressure difference, could be achieved. It is worth mentioning that Ansys-Fluent software is used for simulating and analyzing the problem in this work.

## 2. Materials and Methods

In this section, the mathematical model, that is, governing equations, associated with the understudy system, are presented so that finally it could find an appropriate relation between the fluid flow properties and operational condition of the system. For this objective, momentum and continuity equations, for a 3D turbulent compressible unsteady gas flow through the dampener, are presented. Also,  $k-\varepsilon$  is the chosen turbulent model for modelling the fluid flow in this work.

Continuity equation for the gas flow could be written as follows [34]:

$$\frac{\partial p}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = s_m, \quad (1)$$

where,  $x_i$ ,  $u_i$ ,  $\rho$ ,  $t$ , and  $s_m$  are Cartesian coordinate, velocity vector, fluid density, time, and mass source, respectively. Momentum equation, also, could be given as follow [34]:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j - \tau_{ij})}{\partial x_j} = \frac{\partial p}{\partial x_i} + s_i, \quad (2)$$

where  $p$  and  $s_i$  are pressure and momentum source, respectively.  $\tau_{ij}$  is also tension tensor which is defined as follows [34]:

$$\tau_{ij} = 2\mu s_{ij} - \frac{2}{3}\mu \frac{\partial u_k}{\partial x_k} \delta_{ij} - \overline{\rho u_i u_j}. \quad (3)$$

In the aforementioned equation,  $\mu$  and  $\delta_{ij}$  are fluid viscosity and Kroenker delta, respectively. Strain tensor ( $s_{ij}$ ) could also be given as follows [34]:

$$s_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \quad (4)$$

A very important issue in computational fluid dynamic (CFD) is selecting an appropriate turbulent model. The main objective of a turbulent model is providing a method for calculating the effects of turbulent pulsations on the flow. In this study, the overall properties of the fluid in a closed channel were assessed; therefore, the turbulent model  $k-\varepsilon$  has been chosen to show the effect of turbulence. The turbulence kinetic energy balance could be written as follows [35]:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j} \left[ \rho u_j k - \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] = \mu_t (P + P_B) - \rho \varepsilon - \frac{2}{3} \left( \rho k + \mu_t \frac{\partial u_i}{\partial x_i} \right) \frac{\partial u_i}{\partial x_i} + \mu_t P_{NL}, \quad (5)$$

where  $k$  and  $\varepsilon$  are turbulence kinetic energy and turbulence diffusion rate and the parameters  $P$ ,  $P_B$ , and  $P_{NL}$  are given as follows [35]:

$$P \equiv S_{ij} \frac{\partial u_i}{\partial x_j},$$

$$P_B \equiv - \frac{g_i}{\sigma_{ht}} \frac{1}{\rho} \frac{\partial \rho}{\partial x_i}, \quad (6)$$

$$P_{NL} = \frac{\rho}{\mu_t} \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} - \left[ P - \frac{2}{3} \left( \frac{\partial u_i}{\partial x_j} + \frac{\rho k}{\mu_t} \right) \frac{\partial u_i}{\partial x_i} \right],$$

where turbulence viscosity ( $\mu_t$ ) can be calculated by the following equation [35]:

$$\mu_t = \rho \frac{C_\mu k^2}{\varepsilon}. \quad (7)$$

Turbulence diffusion rate balance is also in the following format [35]:

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} \left[ \rho u_j \varepsilon - \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] = C_{\varepsilon 1} \frac{\varepsilon}{k} \left[ \mu_t P - \frac{2}{3} \left( \rho k + \mu_t \frac{\partial u_i}{\partial x_i} \right) \frac{\partial u_i}{\partial x_i} \right] + C_{\varepsilon 3} \frac{\varepsilon}{k} \mu_t P_B - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + C_{\varepsilon 4} \rho \varepsilon \frac{\partial u_i}{\partial x_i} + C_{\varepsilon 3} \frac{\varepsilon}{k} \mu_t P_{NL}. \quad (8)$$

In this equation,  $C_{\varepsilon 1} = 1.44$ ,  $C_{\varepsilon 2} = 1.92$ ,  $C_{\varepsilon 3} = 1.44$ ,  $\sigma_\varepsilon = 1.22$ ,  $\sigma_k = 1.00$ , and  $C_\mu = 0.09$ .

As it was mentioned earlier, the dampener must reduce amplitude pressure pulsations (A) where,

$$A(\%) = \frac{P_{\max} - P_{\min}}{2P_{\text{line}}} \times 100. \quad (9)$$

In which,  $P_{\max}$ ,  $P_{\min}$ , and  $P_{\text{line}}$  are maximum pressure, minimum pressure in one cycle, and inlet line pressure, respectively. The lowest required volume of dampener in reciprocating compressor suction part has been addressed as follows [32]:

$$V_s = 8.1 \times PD \left( \frac{\gamma T_s}{M} \right)^{(1/4)}, \quad (10)$$

$$V_s \geq 0.03 \text{m}^3. \quad (11)$$

In this equation,  $V_s$ ,  $\gamma$ ,  $T_s$ ,  $M$ , and  $PD$  are respectively the minimum required volume of pulsation dampener, compressibility factor, gas temperature in suction side (K), and molar mass and displaced net volume of cylinders. For calculating  $PD$ , the following equation has been proposed [36]:

$$PD = \left( \frac{\pi}{4} \times D^2 \right) \times L \times N. \quad (12)$$

In which,  $D$ ,  $L$ , and  $N$  are cylinder diameter, stroke length, and number of cylinders, respectively.

The maximum allowable pressure pulsation of dampeners ( $P_1$ ) can be calculated by the following equation [32]:

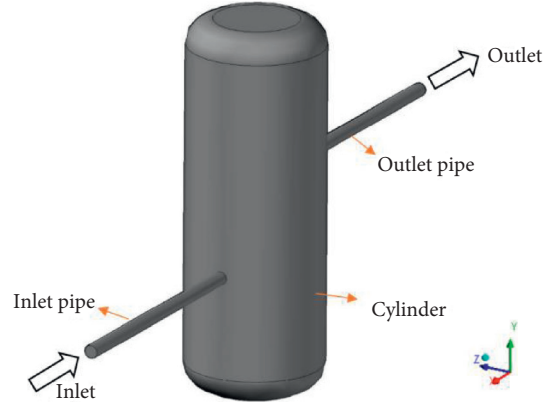


FIGURE 3: A view of geometry and its components.

$$P_1 = \frac{4.1}{(P_{\text{line}})^{(1/3)}} \%. \quad (13)$$

According to this equation, for 17.1 bar pressure line,  $P_1$  is 1.6% of pressure line.

### 3. Numerical Simulation

Here, to simulate the gas flow through the dampener, the computational code of Ansys-Fluent Version 17, by which time-dependent 3D continuity and Navier–Stokes equations by finite-volume method may be calculated, has been used. This code has widely been used for numerical simulation of various complicated geometries with different flow conditions. In this study, the gas flow is considered as an unsteady, compressible, viscous, and Newtonian flow, and the gas itself is assumed pure methane as an ideal gas.

The geometry includes an inlet pipe 2 inches in diameter and 1 metre in length with an outlet pipe 2 inches in diameter in different lengths. In addition, a cylinder has been inserted between the two pipes with different diameters and heights. To find a suitable dampener design, several geometric parameters are investigated. They are the ratio of height to diameter of the cylinder, the length of outlet pipe, and the effect of volume increasing. A view of the geometry and its component is shown in Figure 3. Used geometric layout, parameters location, and placement of plane 1 at the inlet pipe can be seen in Figure 4. The results of the pressure pulsation in this plane at a distance of 10 cm from the cylinder wall are examined. In fact, this plane has been chosen for assessing the pulsation reduction purpose in the dampener.

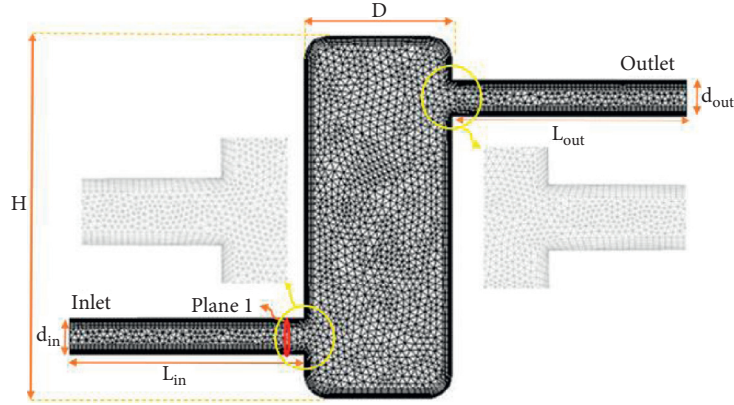


FIGURE 4: Schematic of meshing, dampener's geometric symptoms, and the location of plane 1.

After the geometry has been created, it is time for meshing. Grid quality is essential for solving the right problem. Coarse grid leads to high numerical error, and very small grids make calculations too time-consuming and cumbersome, so finding a suitable mesh is essential. Meshing on the geometry, which is carried out by the same software, is also shown in Figure 4. As can be seen, this computing grid is combination of organization and non-organization mesh. At a distance from the wall, the grid is a pyramid-shaped organization and near the pipe wall and cylinder, the grid is minimized, and a squared organization grid is used. It should be noted that in order to reduce computation time and the number of mesh, a symmetry plane is used in the  $x$ - $y$  plane. For finding a suitable mesh, several meshing is carried out on geometry with taking the dampener's fixed volume equal to the minimum volume, the outlet pressure amplitude of 4% of pressure line and outlet pipe length of 1 m when the height to diameter ratio is 2. Besides, as can be seen in Figure 5, the number of meshes more than 200,000 does not have any effect on the results, which shows that this is the suitable number.

In this study, dampener volume has been investigated for CNG stations with gas inlet pressure of 250 psi. Based on the CNG compressor type, the minimum volume of dampener in the suction is calculated using equation extracted from the standard API 618. Minimum volume size is given in Table 1. The dampener geometry is considered in three dimensions and the solution procedure is totally unsteady as pressure pulsations strongly depend on time. It is assumed that the inlet and outlet temperatures are constant and equal to 303.15 K. Moreover, the process is adiabatic. Employing the data given in Table 1 and considering equations (10) and (11), the minimum value for the required volume of pulsation dampener is obtained as equal to  $0.06 \text{ m}^3$ .

This problem has boundary conditions with constant velocity in inlet plane, pressure pulsation in the output plane, symmetry for applied symmetry plane to the  $x$ - $y$  and finally wall for the cylinder, inlet and outlet pipe walls. Boundary condition in the inlet is constant velocity (velocity inlet), and it is considered in line with the axis of the inlet pipe. According to the type of CNG stations (with input pressure 250 psi), the volume flow rate measured by the

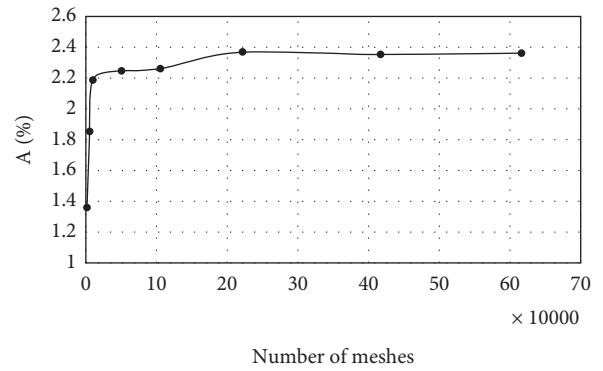


FIGURE 5: An analysis on the number of meshes.

TABLE 1: The minimum cylinder size obtained according to API 618.

Description	Symbol	Unit	Value
Piston displacement in one stroke	PD	$\text{m}^3/\text{rev}$	0.0031
Compressibility factor	$\gamma$	—	1.312
The gas temperature in suction	$T_s$	K	303.1
Molar mass	$M$	$\text{kg}/\text{mol}$	17.57
The minimum volume of dampener	$V_s$	$\text{m}^3$	0.06

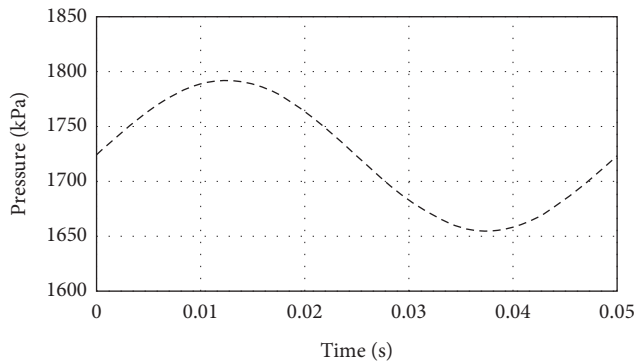
meter turbine in normal conditions is  $950 \text{ m}^3/\text{hr}$ . Owing to the inlet pipe diameter (2 inches) and equaling the mass flow rate of fluid in normal conditions and real conditions, constant speed will be 8.42 m/s on the inlet. It should be noted that condition is normal when the fluid pressure equal to one atmosphere and the temperature is equal to  $15^\circ\text{C}$ , and the condition is real when fluid pressure is 250 psi and temperature is  $25^\circ\text{C}$ . The summary calculation of the fluid velocity in inlet is in Table 2.

Outlet boundary, which is connected to the source of pressure pulsation or compressor, follows the pressure outlet boundary condition. To enter the amount of pressure pulsation as sine time-dependent function, a UDF (user-defined function) is used (Figure 6). The amount of the base pressure is 250 psi that the issue is solved for three ranges of amplitude 5 psi, 10 psi, and 25 psi equivalent to 2%, 4%, and



TABLE 2: The inlet velocity calculated in real condition.

Description	Symbol	Unit	Value
Inlet flow rate in normal condition	$Q_N$	$\text{m}^3/\text{hr}$	950
Methane density in normal condition	$\rho_N$	$\text{kg}/\text{m}^3$	0.7158
The area of the inlet pipe	$A_{\text{in}}$	$\text{m}^2$	0.002027
Methane density in real condition	$\rho_R$	$\text{kg}/\text{m}^3$	11.07
Inlet velocity	$V_R$	$\text{m}/\text{s}$	8.42



--- Outlet boundary condition for 4% of Pressure line

FIGURE 6: Outlet boundary condition.

10% in the pressure line. As the rapidly rotating compressor crankshaft is equal to 1200 r/min, we can see that every compressor cycle happens in 0.5 seconds. Boundary condition at the internal wall of the cylinder and pipes are no-slip condition, meaning that gas adjacent to the border has a zero velocity. This condition is used for solving the equations of continuity, momentum, turbulence kinetic energy and turbulence dissipation rate turmoil of fluid.

Residual values are the best factor for adjusting the convergence criterion in a numerical solution. These residuals are including the remains of the equations of continuity, momentum in the  $x$ ,  $y$ , and  $z$ , energy,  $k$  and  $\epsilon$ . In this study, if the total residues amount for each time step in the equation gets less than  $10^{-5}$ , the convergence condition according to the nonpermanent flow will be satisfied. It should be noted that for converging, maximum 300 repeats in each time step is proposed.

Simulation by dividing the geometry to the small volumes and solving differential equations based on these small volumes, acquires algebraic equations coupled velocity and pressure in the center of each volume. The solutions will start with an initial guess at the pressure for solving the Navier–Stokes equation to achieve the new values of pressure and velocity. This process continues until the convergence criterion is satisfied.

#### 4. Validation of Numerical Simulation

4.1. *A Glance on Former Studies.* Jeong et al. [18] performed a numerical simulation in the hydrogen gas compressors output by using Star CD Code with unsteady flow conditions and  $k$ - $\epsilon$  turbulence model in 2008. Their result was compared with the experimental results of Akbar et al. [37].

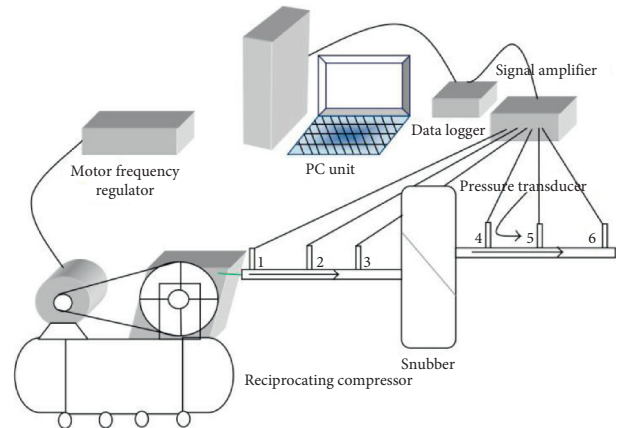


FIGURE 7: Block diagram of experimental setup by Akbar et al. [37].

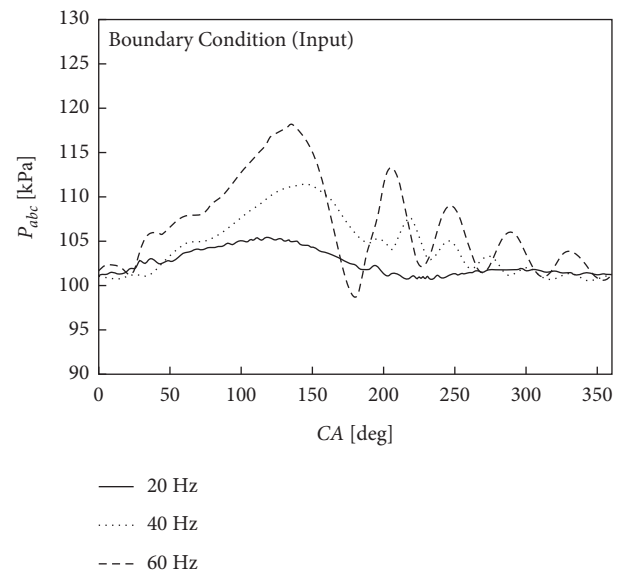


FIGURE 8: Inlet boundary condition for numerical simulation by Akbar et al. [37].

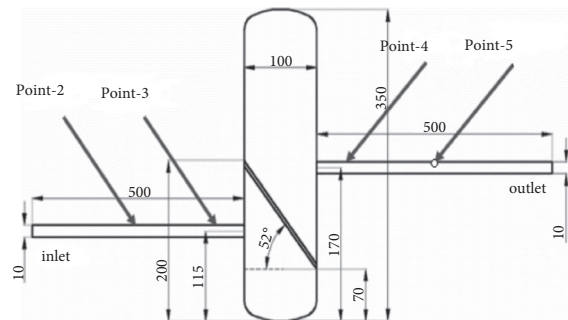


FIGURE 9: Details of dampener dimensions with measuring points by Akbar [37].

Akbar et al. [37] examined a dampener of acrylic with an inlet pipe, an outlet pipe, and a cylinder. Figure 7 presents closed set-up in the foregoing study. Experimental results are obtained for the compressor in the engine speed with 20

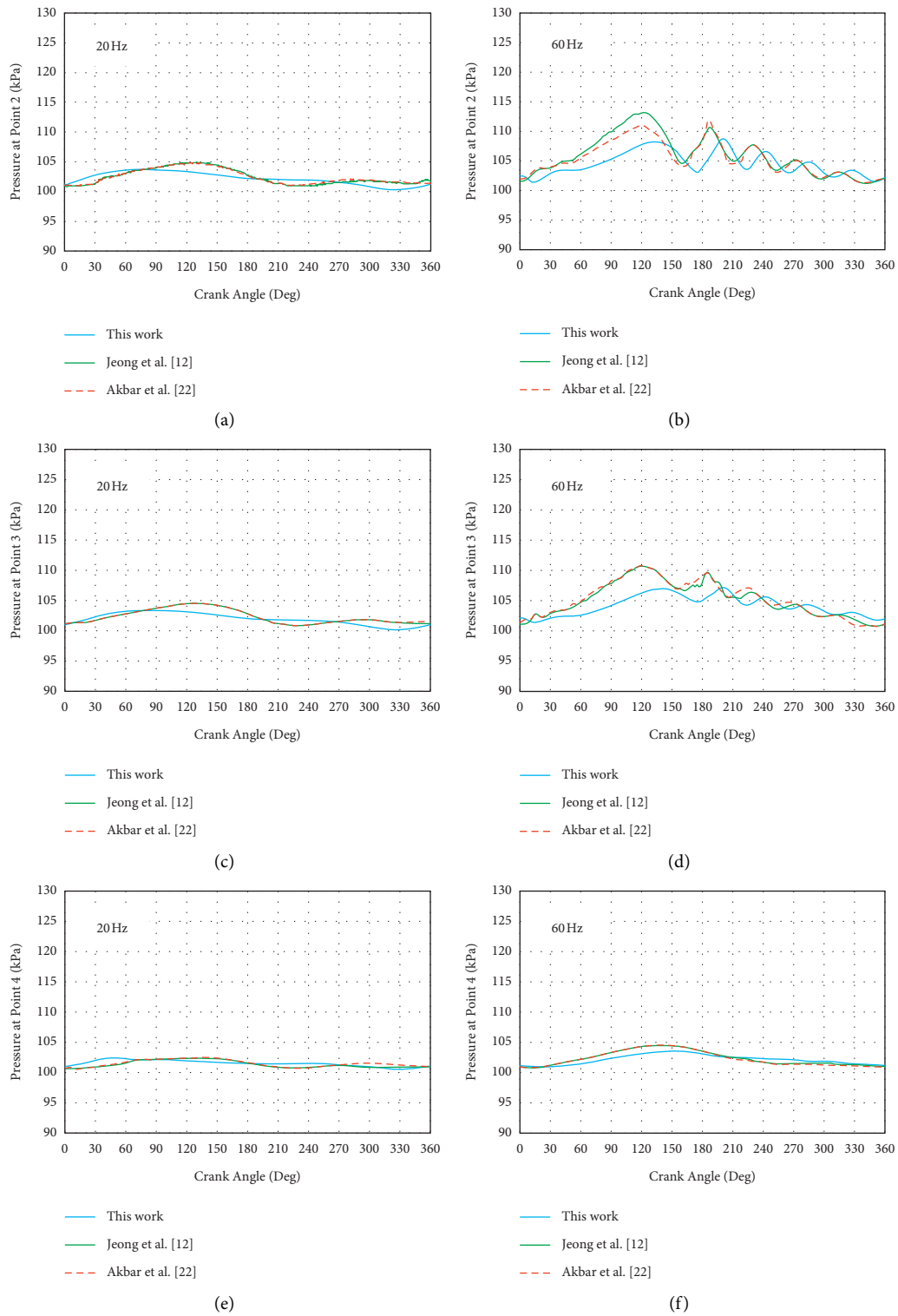


FIGURE 10: Continued.

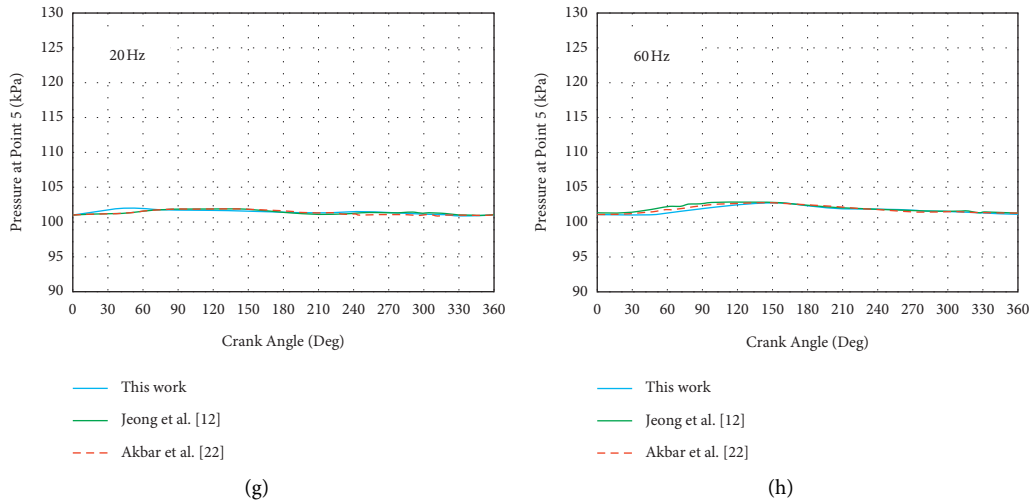


FIGURE 10: Comparison of present study with experimental (Akbar et al.) and numerical (Jeong et al.) works for special case of hydrogen gas with compressor's motor frequency of 20 Hz and 60 Hz, at measurement points 2 to 5. The effects of the length of outlet pipe dampener.

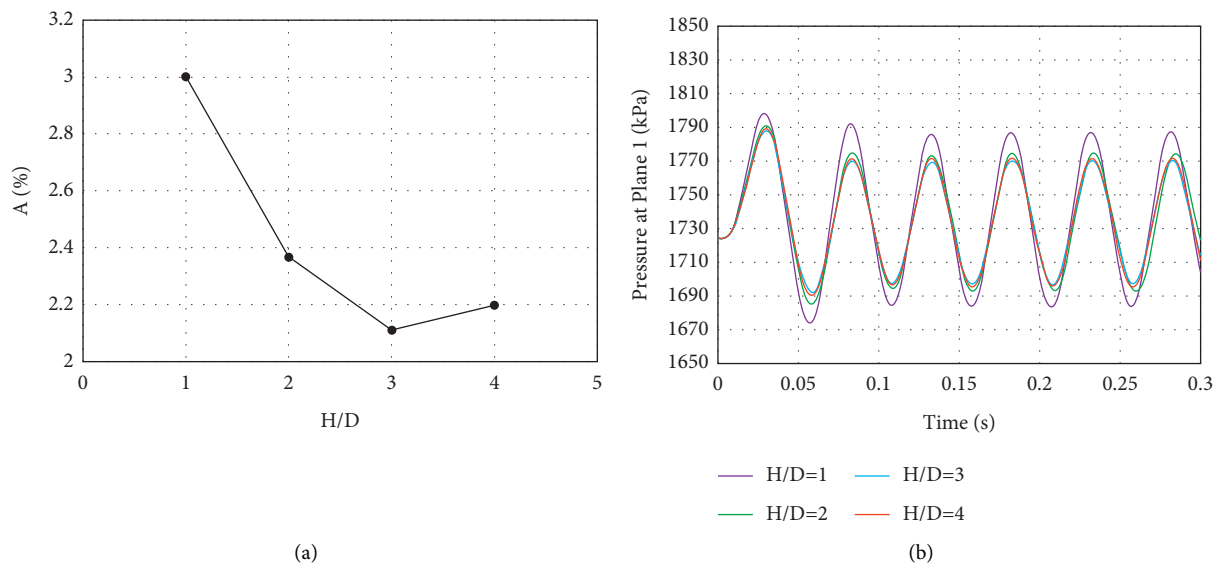


FIGURE 11: (a) The amplitude pressure percent in plane 1, versus  $H/D$  obtained for 4% pressure pulsation in outlet and (b) pressure pulsation in plane 1, versus time for different  $L_{out}$ .

and 60 Hz frequencies. Input boundary condition is shown in Figure 8. They specified four points for measuring the pressure in the input and output pipe, and a flat-screen plate with defined height, width, and thickness in the cylinder at a right angle to the horizontal. Geometry and position of these points are shown fully in Figure 9. Inlet pipe is connected to the compressor and outlet pipe discharges into the atmosphere. Pressure values were recorded on a data logger by pressure sensors installed on the target computer.

**4.2. Verification.** Figure 10 has been plotted to compare our results with previous investigations. The pressure fluctuation changes in all measurement points are in accordance with the aforementioned authorities, and the difference is less

than 2% that shows the validity of the current numerical simulation and next results. Besides, according to Figure 10, as the frequency increases, the amplitude of the pulsation grows. From point 2 to point 5, pulsation decreases, and most of this decrease happens between points 3 and 4.

## 5. Results and Discussions

**5.1. The Effects of the Height to Diameter Ratio of the Dampener Cylinder.** This parameter has been examined in four modes (the ratio of height to diameter of 1, 2, 3, and 4), by taking the dampener's fixed volume equal to the minimum volume, the outlet pulsation amplitude of 4% of pressure line and outlet pipe length of 1 m. API 618 states that for a single-cylinder



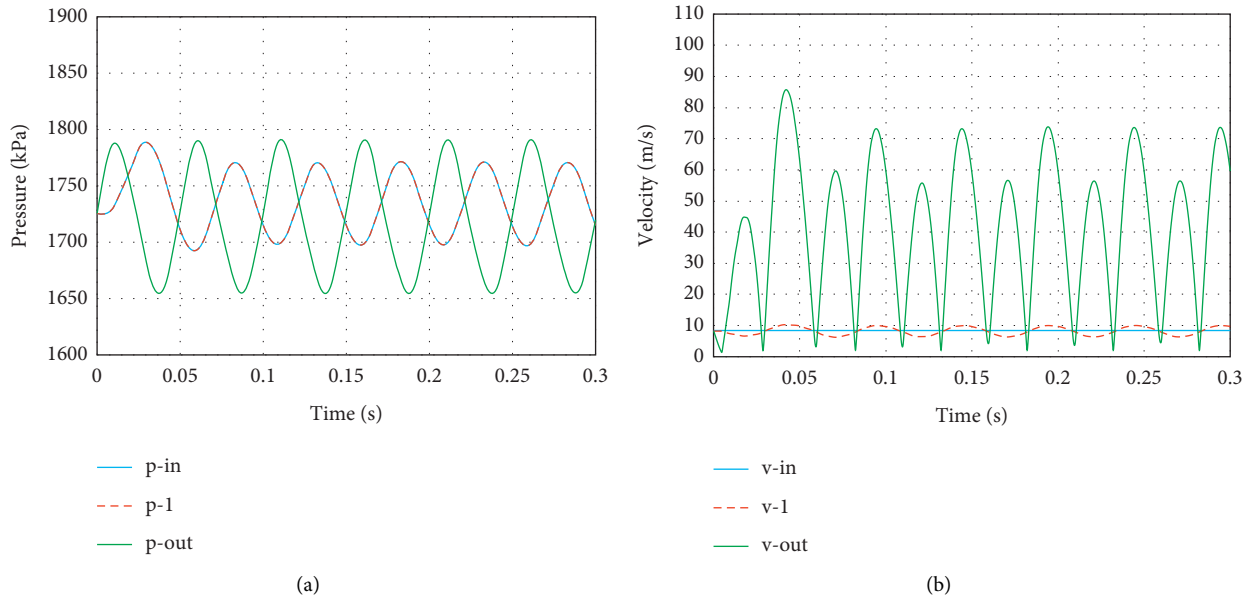


FIGURE 12: The effect of pulsations versus time for  $H/D = 3$ : (a) pressure pulsation and (b) velocity pulsation.

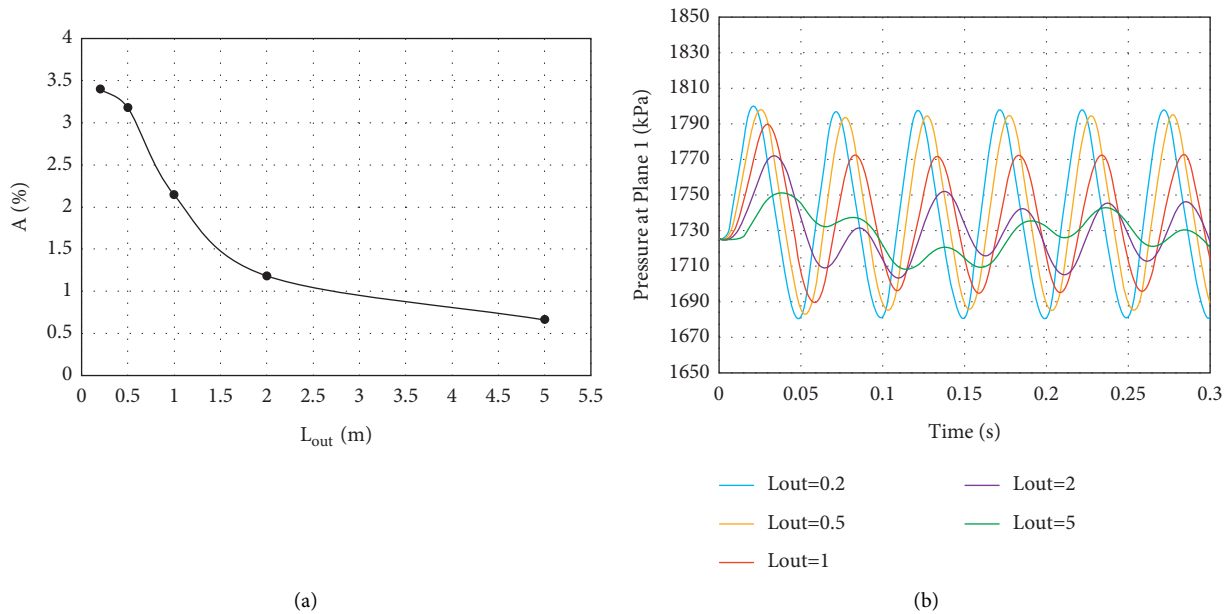


FIGURE 13: (a) The amplitude pressure percent in plane 1, versus  $L_{out}$  obtained for 4% pressure pulsation in outlet and (b) pressure pulsation in plane 1, versus time for different  $L_{out}$ .

dampener, the height to diameter ratio should not exceed greater than 4. As shown in the Figure 11(a), by increasing height to diameter ratio up to 3, the pulsation amplitude on plane 1 will be reduced. After that until  $H/D = 4$ , slightly increases. The minimum pulsation amplitude percent is near 2.1%, which happens when  $H/D = 3$  and this is the best ratio according to the present results. We can find these results by changing pressure over time for six cycles in Figure 11(b). Pressure and velocity changes for inlet, plate 1 and outlet within 0.3 second after the compressor starts to work in  $H/D = 3$  has been presented in Figure 12. As pulsation spends a

little time inside the dampener to reach the inlet, time phase difference occurs at the beginning. The maximum pressure difference between plate 1 and outlet according to figure is 116 kPa. After the second cycle until the compressor works, the pulsation behavior is unchangeable and the maximum and minimum amounts in all cycles are similar.

In fact, the length of outlet pipe dampener ( $L_{out}$ ) is the distance between dampener and compressor or source of pressure pulsation. This parameter has been examined in five modes (the length of outlet pipe dampener of 0.2 m, 0.5 m, 1 m, 2 m, and 5 m), by taking the dampener's fixed volume

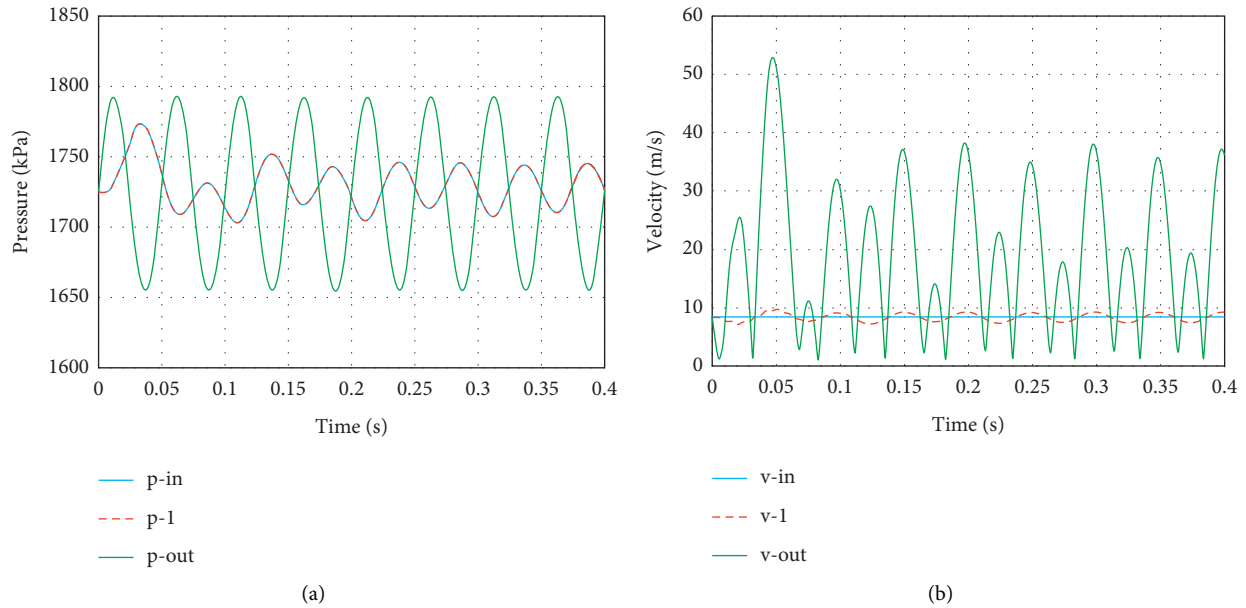


FIGURE 14: The effect of pulsations versus time for  $L_{out} = 2$  m: (a) pressure pulsation and (b) velocity pulsation.

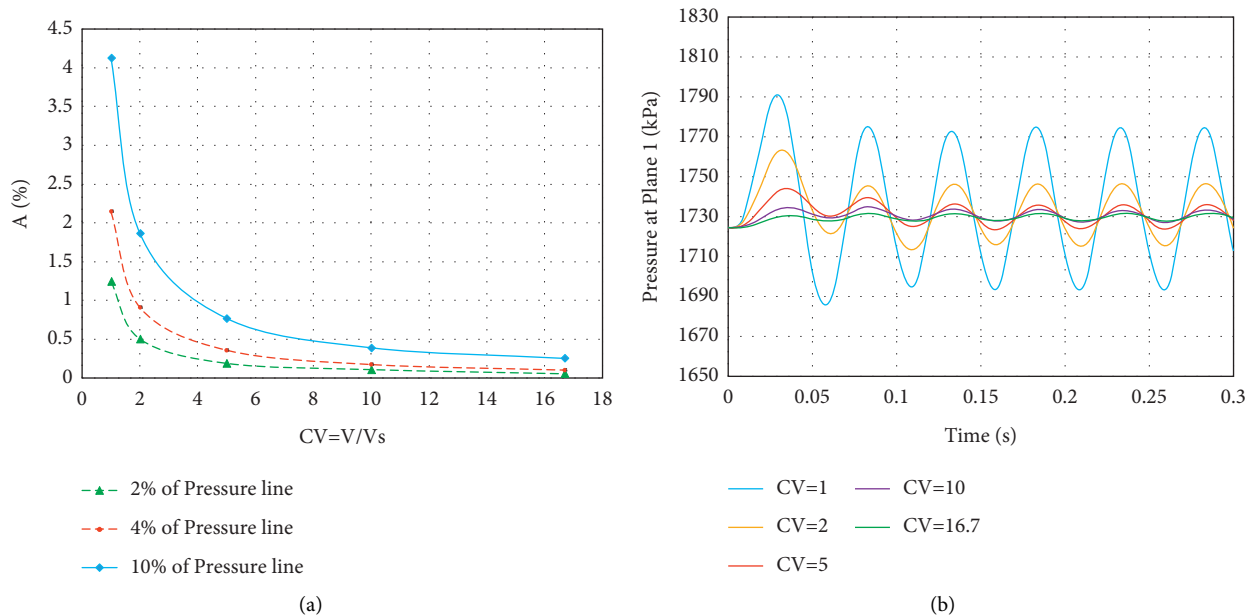


FIGURE 15: (a) The amplitude pressure percent in plane 1, versus CV obtained for 2%, 4%, and 10% pressure pulsation in outlet and (b) pressure pulsation in plane 1, versus time obtained for 4% pressure pulsation in outlet.

equal to the minimum volume, the outlet pulsation amplitude of 4% of pressure line and  $H/D = 3$ . This parameter was investigated to prove that by increasing the distance, pulsation effects on turbine meter could be decreased. Figure 13(a), shows that by increasing the outlet pipe length, the pressure pulsation amplitude on the inlet plane 1 will be reduced, so that this amplitude from 3.4% of pressure line in 2 m length, reaches to 0.7% of pressure line in 5 m length. According to API 618 and equation (13), maximum allowable pressure pulsation of dampeners is 1.6% of pressure line, which happens when the  $L_{out}$  is more than 1.5 m. This result also suggests that pulsation can be significantly

reduced by increasing the distance of the compressor with a small volume equal to minimum volume. We can find these results by changing pressure over time for six cycles in Figure 13(b). The results are not in same phase because pulsation needs more time to affect the inlet of dampener by increasing length. Pressure and velocity changes for inlet, inlet plate 1 and outlet within 0.4 second after the compressor starts to work in  $L_{out} = 2$  m has been presented in Figure 14. After the fourth cycle until the compressor works, the pulsation behavior is unchangeable and the maximum and minimum amounts in all cycles are 1745.5 kPa and 1704.7 kPa, respectively.

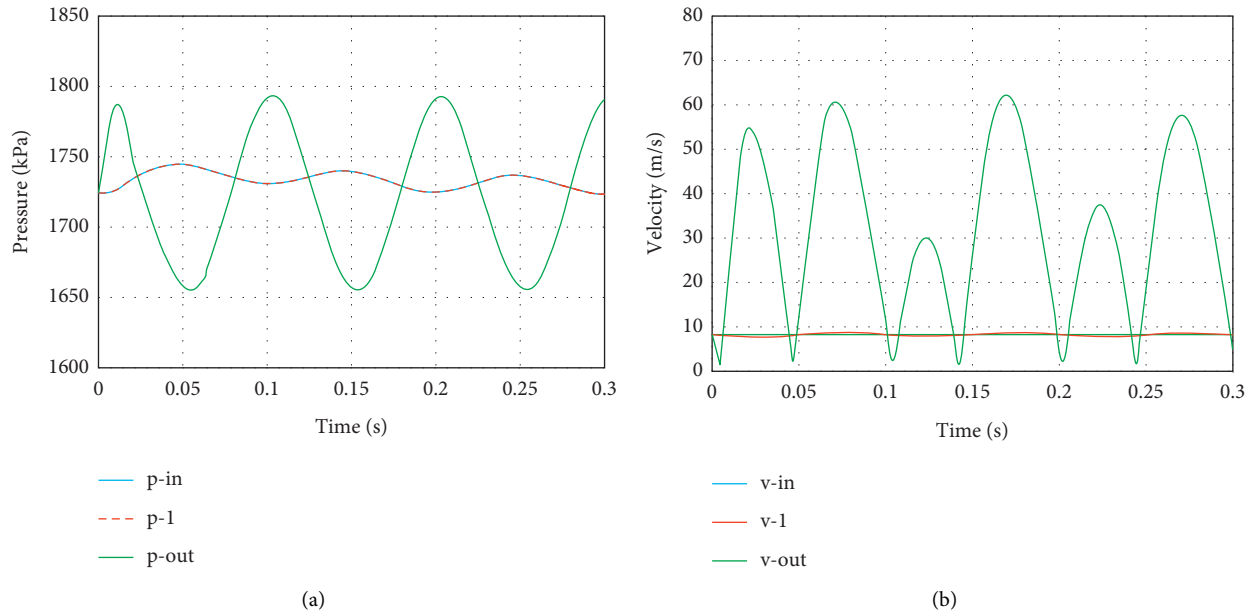


FIGURE 16: The effect of pulsations versus time for CV = 5 and 4% pressure pulsation in outlet: (a) pressure pulsation and (b) velocity pulsation.

**5.2. The Effects of Dampener Volume.** Another important parameter that was analyzed is the effect of the dampener volume increasing in reducing the pressure pulsations. This parameter has been examined in five modes (the volume ratio CV of 1, 2, 5, 10, and 16.7 than the minimum size), by taking the outlet pulsation amplitude of 2%, 4%, and 10% of pressure line,  $H/D=3$  and outlet pipe length of 1 m. Figure 15(a) shows by increasing dampener volume, the pressure pulsation amplitude on the inlet plane 1 will be reduced, while for the outlet pulsation amplitude of 10% of pressure line, this amplitude from 4.1% of pressure line in CV = 1, reaches to 0.25% of pressure line in CV = 16.7. According to equation (13), the maximum allowable pressure pulsation of dampeners is 1.6% of pressure line, which happens when the CV is more than 3. We can find these results by changing pressure over time for six cycles in Figure 15(b). As shown in this figure, after the second cycle until the compressor works, the pulsation behavior is unchangeable and the maximum and minimum amounts in all cycles are similar. The maximum and minimum pressure when CV = 1, respectively, are 1773 kPa and 1695 kPa then their difference is 78 kPa while these amounts for CV = 16.7, are 1731 kPa and 1728 kPa with difference 3 kPa. Pressure and velocity changes for inlet, inlet plate 1, and outlet within 0.3 second after the compressor starts to work in CV = 5 and 4% pressure pulsation in outlet have been presented in Figure 16.

## 6. Conclusion

Axial TGMs are well known and appreciated instruments for custody transfer metering, when used at suitable conditions (steady-flow). In anomalous flow conditions such as pulsating flows or intermittent flows, the

metrological performance of turbine meter can dramatically decay, inducing remarkable systematic errors. In pulsation gas flow, the error of gas turbine is positive because the extra-rotation of the rotor produces an overestimated gas volume. This error makes difference between the amount of gas that have been registered by gas company and sold by CNG station. Therefore, according to API 618, installing appropriate pulsation dampeners at reciprocating compressors inlet for high pressures is vital. Therefore, in the present investigation, an analysis of cylindrical dampener effects on turbine meters' accuracy in a pulsating CNG suction line has been performed. Governing equations modelling was done for compressible fluids, viscous flows, turbulent, and non-slip condition at the walls. Numerical simulation of dampener was investigated by Ansys-Fluent CFD software for a CNG station with the gas inlet pressure 250 psi, inlet and outlet pipe diameter of 2 inches. Our results have been confirmed by good agreement obtained between fluent numerical method with previous studies in a special case. It is worth mentioning, although standard API 618 clearly proposes the lowest required volume and minimum allowable pressure pulsation compared to pressure line for the dampener, it does not give any information about the optimal value of height to diameter ratio ( $H/D$ ) of the device. Besides, for finding optimal volume of dampener, the lowest required volume and minimum allowable pressure pulsation should be investigated before design process. Briefly speaking, this study has acquired the optimal value of  $H/D$  and also the pulsation dampener volume by which the lowest rate of pressure pulsations, at a specific and constant pressure difference, could be achieved. The conclusions drawn from the study can be summarized as follow:

- (i) By increasing height to diameter ratio up to 3, the pulsation amplitude on plane 1 will be reduced. After that until  $H/D = 4$ , slightly increases.
- (ii) The minimum pulsation amplitude percent is near 2.1% that happens when  $H/D = 3$  and this is the best ratio according to present results.
- (iii) As pulsation spends a little time inside the dampener to reach the inlet, time phase difference occurs at the beginning.
- (iv) By increasing the outlet pipe length, the pressure pulsation amplitude on the inlet plane 1 will be reduced 80% in 2 m length respect to 5 m length.
- (v) The numerical results showed 2.4% increase and 1.9% decrease in the maximum and minimum pressure for  $CV = 1$  and  $CV = 16.7$ , respectively.
- (vi) Pressure and velocity changes for inlet, inlet plate 1 and outlet within 0.4 second after the compressor starts to work in  $L_{out} = 2$  m.

## Nomenclature

$A$ :	Amplitude pressure pulsation
$D$ :	Cylinder diameter
$k$ :	Turbulence kinetic energy
$L$ :	Stroke length
$M$ :	Molar mass
$N$ :	Number of cylinders
$P$ :	Pressure
$P_1$ :	Maximum allowable pressure pulsation
PD:	Displaced net volume of cylinders
$P_{line}$ :	Inlet line pressure
$s_i$ :	Momentum source
$s_{ij}$ :	Strain tensor
$s_m$ :	Mass source
$t$ :	Time
$T_s$ :	Gas temperature
$u$ :	Velocity vector
$V_s$ :	Minimum required volume
$x$ :	Cartesian coordinate

## Greek Symbols

$\gamma$ :	Compressibility factor
$\delta_{ij}$ :	Kroenker delta
$\varepsilon$ :	Turbulence diffusion rate
$\mu$ :	Fluid viscosity
$\rho$ :	Fluid density
$\tau_{ij}$ :	Tension tensor

## Subscripts

$in$ :	Inlet
$m$ :	Mass
$max$ :	Maximum
$min$ :	Minimum
$N$ :	Normal condition
$out$ :	Outlet
$R$ :	Real condition

$s$ :	Suction
$t$ :	Turbulence

## Data Availability

The data used to support the findings of this study are included within the article.

## Conflicts of Interest

The authors declare that there are no conflicts of interest regarding the publication of this paper.

## Authors' Contributions

M. F. proposed the numerical simulation and wrote the literature review, the discussions, and the interpretation of the results. S. D. conducted the industrial concepts of work, wrote the verification part, and co-wrote the manuscript. M. F. G. defined the problem, conducted scientific concepts of work, and co-wrote the manuscript. H. T. revised the manuscript and improved the numerical simulation. All authors originated the developed problem and reviewed the manuscript.

## References

- [1] A. Rao, Z. Wu, R. K. Mehra, H. Duan, and F. Ma, "Effect of hydrogen addition on combustion, performance and emission of stoichiometric compressed natural gas fueled internal combustion engine along with exhaust gas recirculation at low, half and high load conditions," *Fuel*, vol. 15, Article ID 121358, 2021.
- [2] A. E. Karac and I. Dincer, "A hybrid compressed natural gas-pneumatic system as a powering option for buses: a comparative assessment," *Energy*, vol. 230, Article ID 120865, 2021.
- [3] Z. Liu, J. Cheng, Q. Feng, and X. Yu, "Effect of a cross-flow perforated tube on pressure pulsation and pressure loss in a reciprocating compressor piping system," *Proceedings of the Institution of Mechanical Engineers—Part C: Journal of Mechanical Engineering Science*, vol. 231, no. 3, pp. 473–484, 2018.
- [4] Q. Ma, Z. Wu, G. Yang, Y. Ming, and Z. Xu, "Pulsation suppression in a reciprocating compressor piping system using a two-tank element," *Proceedings of the Institution of Mechanical Engineers—Part E: Journal of Process Mechanical Engineering*, vol. 232, Article ID 0954408917713436, 2017.
- [5] M. Farzaneh-Gord, M. Faramarzi, M. H. Ahmadi et al., "Numerical simulation of pressure pulsation effects of a snubber in a CNG station for increasing measurement accuracy," *Engineering Applications of Computational Fluid Mechanics*, vol. 13, no. 1, pp. 642–663, 2019.
- [6] H. G. Lee, H. C. Son, H. N. Lee, and G. M. Park, "An experimental study on flow characteristics of turbulent pulsating flow in a curved duct by using LDV," *Transactions of the Korean Society of Mechanical Engineers B*, vol. 25, no. 11, pp. 1561–1568, 2001.
- [7] Y.-T. Yu, G.-D. Na, and J.-H. Kim, "Propagation characteristics of pressure pulse of unsteady flow in a hydraulic pipeline," *Transactions of the Korean Society of Mechanical Engineers B*, vol. 26, no. 1, pp. 1–11, 2002.

- [8] A. R. Al-Obaidi, "Investigation of the influence of various numbers of impeller blades on internal flow field analysis and the pressure pulsation of an axial pump based on transient flow behavior," *Heat Transfer*, vol. 49, no. 4, pp. 2000–2024, 2020.
- [9] E. Mujic, A. Kovacevic, N. Stosic, and I. K. Smith, "The influence of port shape on gas pulsations in a screw compressor discharge chamber," *Proceedings of the Institution of Mechanical Engineers—Part E: Journal of Process Mechanical Engineering*, vol. 222, no. 4, pp. 211–223, 2008.
- [10] H. Wu, Z. Xing, X. Peng, and P. Shu, "Simulation of discharge pressure pulsation within twin screw compressors," *Proceedings of the Institution of Mechanical Engineers—Part A: Journal of Power and Energy*, vol. 218, no. 4, pp. 257–264, 2004.
- [11] Z.-D. Qian, J.-D. Yang, and W.-X. Huai, "Numerical simulation and analysis of pressure pulsation in Francis hydraulic turbine with air admission," *Journal of Hydrodynamics*, vol. 19, no. 4, pp. 467–472, 2007.
- [12] K. N. Atkinson, "A software tool to calculate the over-registration error of a turbine meter in pulsating flow," *Flow Measurement and Instrumentation*, vol. 3, no. 3, pp. 167–172, 1992.
- [13] K. W. Chau and Y. W. Jiang, "A three-dimensional pollutant transport model in orthogonal curvilinear and sigma coordinate system for pearl river estuary," *International Journal of Environment and Pollution*, vol. 21, no. 2, p. 188, 2004.
- [14] A. R. Al-Obaidi, "Monitoring the performance of centrifugal pump under single-phase and cavitation condition: a CFD analysis of the number of impeller blades," *Journal of Applied Fluid Mechanics*, vol. 12, no. 2, pp. 445–459, 2019.
- [15] A. R. Al-Obaidi and H. Towsyfyhan, "An experimental study on vibration signatures for detecting incipient cavitation in centrifugal pumps based on envelope spectrum analysis," *Journal of Applied Fluid Mechanics*, vol. 12, no. 6, pp. 2057–2067, 2019.
- [16] G. Vetter and F. Schweinfurther, "Pressure pulsations in the piping of reciprocating pumps," *Chemical Engineering & Technology—CET*, vol. 10, no. 1, pp. 262–271, 1987.
- [17] S. Notzon, "Low pulsation design of piping systems for high pressure reciprocating pumps," *Process Technology Proceedings*, vol. 12, pp. 575–580, 1996.
- [18] H. M. Jeong, H. S. Chung, W. A. Akbar, G. H. Lee, K. J. Shim, and Y. H. Lee, "Numerical analysis on various models of pressure snubbers in the hydrogen gas compressing system," *Journal of Mechanical Science and Technology*, vol. 22, no. 4, pp. 761–769, 2008.
- [19] A. R. Al-Obaidi, "Effects of different turbulence models on three-dimensional unsteady cavitating flows in the centrifugal pump and performance prediction," *International Journal of Nonlinear Sciences and Numerical Simulation*, vol. 20, 2018.
- [20] A. R. Al-Obaidi and A. A. Mohammed, "Numerical investigations of transient flow characteristic in axial flow pump and pressure fluctuation analysis based on the CFD technique," *Journal of Engineering Science and Technology Review*, vol. 12, no. 6, pp. 70–79, 2019.
- [21] A. R. Al-Obaidi, "Numerical investigation of flow field behaviour and pressure fluctuations within an axial flow pump under transient flow pattern based on CFD analysis method," *Journal of Physics: Conference Series*, vol. 1279, no. 1, Article ID 012069, 2019.
- [22] A. R. Al-Obaidi, "Experimental investigation of cavitation characteristics within a centrifugal pump based on acoustic analysis technique," *International Journal of Fluid Mechanics Research*, vol. 47, no. 6, Article ID 2020029862, 2020.
- [23] A. R. Al-Obaidi, "Experimental comparative investigations to evaluate cavitation conditions within a centrifugal pump based on vibration and acoustic analyses technique," *Archives of Acoustics*, vol. 45, no. 3, pp. 541–556, 2020.
- [24] M. Farzaneh-Gord, A. Niazmand, M. Deymi-Dashtebayaz, and H. R. Rahbari, "Effects of natural gas compositions on CNG (compressed natural gas) reciprocating compressors performance," *Energy*, vol. 90, pp. 1152–1162, 2015.
- [25] D. Świsulski, R. Hanus, M. Zych, and L. Petryka, "Methods of measurement signal acquisition from the rotational meter for frequency analysis," *EPJ Web of Conferences*, vol. 143, Article ID 02124, 2018.
- [26] J. Slůse, R. Maury, A. Andreas, F. Olsen, J. Geršl, and A. Kenbar, "Influence of upstream disturbances on performance of an LDV-based cryogenic flow meter standard—CFD modelling and preliminary measurements with air," *Flow Measurement and Instrumentation*, vol. 80, Article ID 101991, 2021.
- [27] M. L. Ali, R. Ridoy, U. Barua, and M. B. Alamgir, "Design and fabrication of a turbine meter," *Journal of Modern Science and Technology*, vol. 4, no. 1, pp. 16–26, 2016.
- [28] ISO 9951, *Measurement of Gas Flow in Closed Conduits Turbine Meters*, International Organization for Standardization, Geneva, Switzerland, 1993.
- [29] S. Y. Sun, K. Cheng, T. Ren, and S. K. Yang, "Studies on the pressure pulsation of plenum chamber in reciprocating compressor using recognition technique," in *Proceedings of the International Compressor Engineering Conference*, vol. 867, pp. 751–758, Purdue University, West Lafayette, IN, USA, 1992.
- [30] F. Cascetta and G. Rotondo, "Effects of intermittent flows on turbine gas meters accuracy," *Measurement*, vol. 69, pp. 280–286, 2015.
- [31] F. Cascetta, G. Rotondo, and M. Musto, "Measuring of compressed natural gas in automotive application: a comparative analysis of mass versus volumetric metering methods," *Flow Measurement and Instrumentation*, vol. 19, no. 6, pp. 338–341, 2008.
- [32] Standard API 618, *Reciprocating Compressors for Petroleum, Chemical, and Gas Industry Services*, American Petroleum Institute, Washington, DC, USA, 5th edition, 2007.
- [33] A. Maekawa, T. Tsuji, M. Noda, T. Takahashi, M. Kato, and K. Fujita, "Experimental and numerical study on pressure pulsations under various acoustic boundary conditions in piping systems," *Journal of Pressure Vessel Technology*, vol. 139, no. 2, Article ID 021703, 2017.
- [34] F. M. White, *Viscous Fluid Flow*, McGraw-Hill, New York, NY, USA, 3rd edition, 2006.
- [35] S. B. Pope, *Turbulent Flows*, Cambridge University Press, Cambridge, UK, 2000.
- [36] E. W. McAllister, *Pipeline Rules of Thumb Handbook: A Manual of Quick, Accurate Solutions to Everyday Pipeline Engineering Problems*, Gulf Professional Publishing, Houston, TX, USA, 2013.
- [37] W. A. Akbar, K. J. Shim, and C. S. Yi, "Gas pressure fluctuation characteristics inside pipe line passing through a snubber for hydrogen gas compressor," in *Proceeding of the International Conference on Sustainable Energy Technologies*, vol. 30, pp. 659–664, Vicenza, Italy, January 2006.