Experimental and Numerical Investigation of Vortex-Induced Vibrations and Pressure Drop in Square Pipe with Obstacle

Kadhum Audaa Jehhef\textsuperscript{1}, Mohamed Abed Al Abas Siba\textsuperscript{2}

\textsuperscript{1}Department of Mechanical Power, Institute of Technology, Middle Technical University, Iraq.
\textsuperscript{2}Department of Mechanical Power, Institute of Technology, Middle Technical University, Iraq.

ABSTRACT: In this paper, a study on the fluid vortex-induced vibration caused by an obstacle used as vortex generator inserted in a square pipe is conveying a water flow with laminar and turbulent flow condition has been considered. The water flows in the pipe with length of L=1000 mm and side length a=20 mm. The obstacle located in the middle section of the duct with height to duct side ratio given by (h/a) as 0.0, 0.25, 0.5 and 0.75. The flowrate of the water that enter to the pipe with Reynolds number that varied by 860, 1300, 1700 and 2100 for the laminar flows and with range of 3500, 430, 5200 and 6900 for turbulent flow. The time history signals of the structure vibration accelerations measured by the accelerometer. Simulation is carried out by using ANSYS-FLUENT to solve the Navier-Stokes equation by adopting $k$-$\varepsilon$ turbulence model. The experimental results are given by natural frequency of the free vibration signals of the pipe structure of the test section. The results indicated that the using an obstacle lead to increase the pipe frequency. As a result, flow induced vibration is also observed at moderately in high Reynolds numbers. The results obtained by experimental tests indicated there is a strong effect of the Reynolds number on the vibration signal. These vibration signals are believed to be caused by friction coupling between the fluid and the wall pipe in the case of smooth pipe and by vortex generated due to the obstacle. The frequency is the highest values at the case of high ratio of obstacle due to high fluid velocity and pressure changes across the obstacle. Finally, the results show that the obstacle heights have a significant effect on the pressure drop.

KEYWORDS: Square pipe Vortex-Induced Vibrations, Obstacle.

INTRODUCTION

The applications of the vortex-induced vibration (VIV) were found in many industrial applications structures such as piping systems, spar platforms, pipelines, and risers. There is a growing need to develop analytical methods for numerical analysis describing this phenomenon. The reliable estimation of magnitude and presence of displacements and excitation modes in response of the VIV to various axial tensions and current profiles presented in many practical fields to the industry. For example, the VIV is a major cause of the failure of fatigue in many pipeline structures [1]. Liquid with external objects or impurities flowing in the pipes causes gradual built-up on the pipe surface area and water pipes under this condition since the nucleation of ice occurs between -4 to -6°C [2]. Due to internal pressure change, clogging could cause the pipe to burst, in some cases. There is a significant importance of the dynamic behavior of the fluid-conveying pipes vibration for many industry applications such as petrol and gas transportation systems [3, 4].

Many pipe systems sometimes subjected to vibration and would be cracked as a result of various external or internal forces due to conveying fluid. Identifying blockage or obstacle in pipe was presented by a number of researches, to ensure the quality of medium. However, the effect of the velocity of critical flow in parameters of the open crack on the fluid-conveying vibration behavior cracked pipe surrounded by a medium of visco-elastic was studied [5]. Their results showed that the presence the pipe crack decreases the critical flow velocity and the natural frequency. To determine blockage in duct via single measurement of transfer function, the rapid spectrum method [6]. Also, presented a study to evaluate the effects of blockages on the nuclear power plant pressure transmitter sensing lines [7]. To study the correlation of blockage levels to vibration signal, investigated the blockage effects in circular pipe by using the measurement of vibration [8]. The streamlines get closer when fluid flow through an obstacle. The results showed that the obstacle will decrease the pressure and increase the flow
velocity. In almost applications of buildings, pipes employ a convey fluid in pipes to a desired location. Liquid containing impurities or foreign objects will sometimes create clog obstruction or unintentional built up along the interior pipe wall.

The vibration measurement to analyze the different blockage sizes impact inside a clear Polyvinyl Chloride (PVC) circular pipe was used [9]. The cracks caused by vibration-based detection were studied [10]. By using the single point on the component, it can determine the size and location of a crack. To predict such length, many efforts have been made recently such as in [11]. Also, a studied experimentally and theoretically the detection of single cracks in pipes filled with fluid under pressure [12]. However, a prediction VIV model of the 3D long flexible pipe had been predicted based on phenomenological VIV model was analyzed by finite difference method (FDM) [13]. Recently, the responses of the flow-induced vibration (FIV) of pipe conveying internal fluid flow was studied by many literatures in the last few decades [14-16]. Furthermore, the nonlinear dynamics 3D VIV model for a flexible fluid-conveying pipe were performed [17]. In addition, constructed a pipe transporting two-phase fluid flow model by using the Timoshenko beam theory [18,19]. The objective of this study is to analysis and investigate experimentally and numerically the effect of presence an obstacle attached to the upper wall of the square duct on the vortex-induced vibration (VIV) dynamic responses. Therefore, this present work studied the effect the of inlet Reynolds number for laminar and turbulent flow conditions and the obstacle height effects on the vibration fundamental conditions.

EXPERIMENTAL INVESTIGATION

The experimental rig schematic is utilized in this study is illustrated in Figure 1. The closed loop of 20 mm-square cross section aluminum pipe, 1-mm wall thickness and 1000-mm long used in this experimental. At the upstream of the square pipe is driven by a centrifugal pump to turn the water between in the experimental rig test section as closed loop water flow. The flow controlled by a throttling valve and the water flow rate that entered the test section is measured by a rotameter. The Reynolds number is varied by 860, 1300, 1700 and 2100 for laminar flow and is varied by 3500, 4300, 5200 and 6900 for turbulent flow. Two calibrated pressure transducer that connected with a digital screen is used to measure the pressure drop across the obstacle. The simply supported condition of the pipe given by two fixed wooden supported was used to fix the pipe from its both ends. The present experimental tests are performed in order to observe the fluid effects on obstacle vibrations and stability of simply supported pipe supports at different fluid velocities. The obstacle height to square pipe side ratio \( h/a \) is varied by 0, 0.25, 0.5, and 0.75. At each \( h/a \), the flow induced frequencies are obtained by placing the accelerometer above the obstacle portion of the square pipe, while the other end of the sensor is connected to a computer loaded with software that analyzes and records online frequency data such as the displacement, velocity and acceleration of the vibrated wall pipe. The displacement measurements are determined experimentally by vibrational analyzer (SKF-El-Kalci). The total time of each run was set at 90 s at interval of 0.145 ms for the experimental setting. The experimental data used was obtained using the fast Fourier transformation (FFT) analyzer and test runs were conducted on a linearly restrained pipe conveying fluid.
Experimental and Numerical Investigation of Vortex-Induced Vibrations and Pressure Drop in Square Pipe with Obstacle

Figure 1. Experimental rig schematic of the square pipe conveying water.

NUMERICAL SOLUTION

In the present work, the illustration geometry configuration of the physical problem of the square pipe is performed in Figure 2. The length of the square pipe is fixed at 1000 mm and the side length is fixed at 20 mm. In the numerical simulations, 2-D of the square pipe is built by using solid-work v.014. the boundary conditions of the flow consist of the inlet water velocity is chosen for laminar flow by 0.04, 0.06, 0.08 and 0.12 m/s and for turbulent flow is varied by 0.17, 0.21, 0.25 and 0.33 m/s. Moreover, at the outlet, the boundary condition set as 0 Pa for gage pressure.

Figure 2. Schematic of the square pipe test section dimensions.

In the numerical solution part, the computational domain is generated by the quad mesh by using (73×270) rectangular cells. In this study, there are three different computational meshes intervals are 0.1 mm, 0.5 mm and 1 mm. The steady state problem, the numerical solution is depended upon the pressure based solver as a default solver setting to solve and utilizing the second order upwind scheme based for the governing equations in this work and these equations are solved by using the finite-volume method based on the ANSYS-FLUENT v.16.0 package. Conservation of momentum in the pipe is described by. The following equations presented and solved in differential form as FLUENT, [20, 21]:

Continuity equation:
\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_M
\]

The momentum given by:
\[
\frac{d}{dt}(\rho \cdot \dot{v}) + \nabla \cdot (\rho \dot{v} \cdot \dot{v}) = -\nabla P + \nabla \cdot \tau + \rho \ddot{g} + \ddot{F}
\]  
(2)

The two equations model is simplest and famous turbulence model. For the solution of many practical engineering flow problems, the standard \( k - \varepsilon \) turbulence model has become the widely used. The standard \( k \) equation of transport for given \([22]\):

\[
\frac{d}{dt}(\rho \cdot k) + \frac{d}{dx_i}(\rho k u_i) = \frac{d}{dx_j}\left[\left(\mu + \frac{\mu_t}{\varepsilon} \right) \frac{dk}{dx_j}\right] + G_k + G_b - \rho \varepsilon - Y_M + S_k
\]  
(3)

The transport equation for the standard \( \varepsilon \) model given by:

\[
\frac{d}{dt}(\rho \cdot \varepsilon) + \frac{d}{dx_i}(\rho \varepsilon u_i) = \frac{d}{dx_j}\left[\left(\mu + \frac{\mu_t}{\varepsilon} \right) \frac{d\varepsilon}{dx_j}\right] + C_1\varepsilon \frac{\varepsilon}{k} \left(G_k + C_{ee}G_b\right) - C_2\varepsilon \frac{\varepsilon^2}{k} + S_\varepsilon
\]  
(4)

The modeling of the turbulent viscosity formed as:

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}
\]  
(5)

The turbulent model constants used in this study are \([22]\):

\[
C_1=1.44C_2=1.92C_\mu=0.09 \sigma_1=1.0 \quad \sigma_\varepsilon=1.3
\]

RESULTS AND DISCUSSION

Displacement Analysis

The square pipe that conveying water is considered in this study, it is used to investigate the influence of varying the obstacle height ratio \((h/a)\) and the Reynolds number \((Re)\) in a simply supported square pipe on the vortex-induced vibrations, friction factor and pressure drop. The experimental results obtained by using the pipe wall displacement and FFT of the natural frequency of the test segment. However, algorithm considers the root-mean square values \((rms)\) rather than the average is employed and stored in MATLAB to conduct the FFT analysis. The technique is based on signal noise from an accelerometer attached to the surface of the pipe above the obstacle region. The wall displacement is plotted in Figures 3 to 6. The results showed that the data is symmetric around the “0” level which means the average of the displacement over the period of the run is zero. Figures 3 and 4 presented the effect of increasing the inlet Re for laminar and turbulent flow upon the pipe wall displacement, the results show that the increasing in the Reynolds number will increase the wall displacement values and the pipe frequency for the case of the without obstacle \((h^*=0.0)\).

Using an obstacle inside the pipe will lead to increase the pipe frequency and pipe vibration displacement with increasing the water flow rate inside the pipe as shown in Figure 5. Typical data obtained in each run at the obstacle height ratio chosen as \((h/a = 0.25, 0.50, and 0.75)\) and Reynolds number at \(Re=850\). The data are discrete due to the nature of gathering the data by proposing a time interval chosen to run each trial. The results indicated that the increasing in the obstacle height ratio will lead to increase the wall displacement in the time domain as demonstrated experimentally as measurement by vibration analyzer, due to the vortex generated behind the obstacle. The displacement results shown in Figure 6 suggest three outcomes: First, the pipe shows very small displacement at low Reynolds number and as Reynolds number increases, the displacement increases. The second outcome is that the pipe shows maximum displacement at the location of the obstacle due to the maximum impact of the flow which appears at the obstacle location \((l=0.5m)\). The third outcome is that at highest obstacle height ratio of \((h^*=0.75)\), the highest displacement appears. The results showed a strong relationship between the water mass flow rate in the pipe and vibration signal generated by the pipe wall. This phenomenon is believed to be happened by friction coupling between the flow of the fluid and the wall of the pipe in the case of smooth pipe and by vortex generated due to the obstacle. And the frequency is the highest in the case of high fluid velocity and pressure changes in the region near the obstacle.
Experimental and Numerical Investigation of Vortex-Induced Vibrations and Pressure Drop in Square Pipe with Obstacle

Figure 3. Pipe displacement results for obstacle height ratio of $h/a = 0.0$ and for various laminar Reynolds numbers.

Figure 4. Pipe displacement results for obstacle height ratio of $h/a = 0.0$ and for various turbulent Reynolds numbers.
In the condition of the laminar and turbulent water flow, the restriction of fluid flow by the obstacle in the present square pipe will cause a continuous vibration and increase the wall displacement. Therefore, using the vibration analysis, the relationship between obstacle height ratio inserted in square aluminum pipe and parameters of the vibration were summarized in Figures 7 and 8. The obstacle restriction will reduce the area of the flow in the square pipe and then increase the water velocity and reduce the pressure in the region that behind the obstacle. Thus, there is a vortices were generated in this region and that cased a pipe wall vibration.

![Figure 7](image)

**Figure 7.** Effect of obstacle height ratios on the pipe displacement results for laminar Reynolds number.

---

(c) $h/a = 0.75$

**Figure 5.** Pipe displacement results for various obstacle height ratios and for laminar Reynolds number of 850.

(a) $h/a = 0.25$

(b) $h/a = 0.5$

(c) $h/a = 0.75$

**Figure 6.** Pipe displacement results for various obstacle height ratios and for turbulent Reynolds number of 7000.
Experimental and Numerical Investigation of Vortex-Induced Vibrations and Pressure Drop in Square Pipe with Obstacle

Figure 8. Effect of obstacle height ratios on the pipe displacement results for turbulent Reynolds number.

Fast Fourier Transform (FFT) Analysis

The frequency of either experimental data is obtained directly from FFT analysis with the help of MATLAB. The trend of the frequency could be viewed at two different perspectives: one when increasing the obstacle height ratio at given Reynolds number at Re=7000 while the other is performed as the Reynolds increases at a given an obstacle height ratio $h^*=0.5$. Figures 9 and 10 plotted the values of RMS of the displacement in a frequency domain. The peaks in the figures represent the estimated values of the frequencies which are shown at 2.5, 12, 17, 25, 32, and 42 Hz. In the first trend where the vibrational frequencies are estimated at a given Reynolds number and increasing obstacle height ratio from 0.0 to 0.5, the higher frequencies of 53 Hz and 45 Hz and others disappear as the aspect ratio increases. This behavior could be interpreted in conjunction with the decreasing of the flow velocity passed the obstacle due to a given fixed Reynolds number. The second trend of flow-induced vibrational can be shown in Figure 10 where measurements are taken at a given obstacle height ratio (0.5) while Reynolds increases from 850 to 7000 (all turbulent). The positions of the vibrational frequencies showed significant changes when the Reynolds number increased to 7000. Seemingly, the results suggest that all the turbulent Reynolds number values employed in this study are capable to produce flow-induced frequencies as shown in the figure.
**Figure 9.** The vibrational frequency of FFT transform signal with frequency domain at a given Reynolds number $Re=7000$.

![Graph](image)

b) $Re=850$

![Graph](image)

b) $Re=7000$

**Figure 10.** Variation of the vibrational frequency of FFT transform signal with frequency domain at a given obstacle height ratio of $h^*=0.5$.

Pressure drop and Friction factor

Practically, vortices occur in the region that behind the obstacle, and this phenomenon will lead to generate an eddy, due to a sharp static pressure drop in the region downstream of the obstacle plate. The results indicated that the values of the pressure level maximum fluctuating are appeared at the obstacle height ratio of $(h/a=0.75)$. Here, $a$ represents side length of square pipe. The results given by Figure 11 show that the obstacle heights have a significant effect on the pressure drop.
In this study, the pressure drop in a square pipe for turbulent flow was described by the relation of the friction factor \( f \) given by [23]:

\[
f = \frac{2\Delta P D_h}{\rho u^2}
\]

(6)

To investigate the effect of Reynolds Number on turbulence intensity and friction factor, the experimental results were plotted in Figure 12. The figure illustrate relationship between the increasing the Reynolds number on the flow friction factor. The results included that the any increasing in the inlet \( Re \) will cause to increase the wall friction factor in the range of \( Re=100 \) to 1800 but the friction factor decreases as the Reynolds number increasing for \( Re \) larger than 2000 with good agreement as compared with that friction factors data of [24].

![Figure 11](image1.png)

**Figure 11.** Pressure drop inside the square pipe variation with the inlet Reynolds number.

![Figure 12](image2.png)

**Figure 12.** Variation of the friction factor with the inlet Reynolds number with different obstacle height ratio.

**Numerical Velocity and Vorticity**

In the numerical results part, the stream line of the velocity and vorticity of the secondary flow that generated by using the obstacle inside the square pipe are investigated according to the results that given by ANSYS-FLUENT data. The main outcome that ANSYS is provided in this study is the velocity profile of the stream line along the
axial dimension of the testing pipe whose characteristics are mentioned in chapter three. As shown at the central line in Figure 13, the vectors showed that the water flow is smoothly in the region of the obstacle downstream without any fluctuating and disturbance suggesting that the effect of the fluid-structure interaction (FSI) is very minimal. However, in the region in the upstream stream line, the flow shows circulation cell which extends to the structure (pipe wall) where the effect of FSI. The appearance of the circulate behind the obstacle on (vorticity) suggests creation of force of viscous which has signifying effect at the pipe inner surface. The force of viscous is the outcome of FSI which signifies the occurrence of the pressure and at the piping wall. This pressure creates stress, strain, and deformation of the pipe structure. In the downstream region where vortices are formed and the shape of the vorticity is varying with increasing the Re and/or with the obstacle height ratio increases. The change in the vorticity geometrical shape suggests variety of coupling between the interaction of the fluid and the structure. The results indicated that the at high Reynolds number of 7000, the vorticity is formed very close to the obstacle while at low Reynolds number of 850, a small vorticity will generate near the obstacle by generating a region of the low pressure. Increasing the obstacle height to (h*=0.5) will lead to increase the diameter of the circulation cell in the obstacle downstream as presented in Figure 14.
b) \( Re=7000 \)

**Figure 13.** The flow stream velocity across the obstacle with obstacle height ratio of \( h^* = 0.25 \) with different Reynolds number.

a) \( Re=860 \)
CONCLUSIONS

An experimental and numerical work to investigate fluid vortex-induced vibration that caused by using an obstacle inside a simply supported square pipe conveying a water flow with turbulent and laminar flow condition were investigated in this work experimentally and numerically. The results concluded that the increasing in the Reynolds number will increase the wall displacement values and the pipe frequency for the case of the without obstacle ($h^*=0.0$). Also, the results indicated that the increasing in the obstacle height ratio will lead to increase the wall displacement in the time domain as demonstrated experimentally as measurement by vibration analyzer, because the vortex generated in the behind the obstacle location. The highest obstacle ratio ($h^*=0.75$), the highest displacement appears, and the frequency is the highest in the case of high fluid velocity and pressure changes in the region near the obstacle. Finally, the results concluded that the insertion an obstacle in the square pipe lead to increase the pressure drop and the friction factor with high percent.

NOMENCLATURES

- $a$: length of square pipe side, m
- $h$: height of the obstacle, m
- $h^*$: obstacle height ratio, ($h^*=h/a$).
- $D_h$: hydraulic diameter, m
- $P$: perimeter of square pipe, m
- $A$: square pipe area, $m^2$
- $u$: water inlet velocity, $m/s$
- $p$: dimensional pressure, $N/m^2$
- $Re$: Reynolds number, $Re = u_iD_h/v$.
- $x, y$: Cartesian coordinates, m
- $g$: acceleration of gravitational, $m/s^2$
- $L$: square pipe length, m
- $S_l, S_c$: source terms
- $\Delta P$: Pressure drop across the obstacle, $pa$
Greek Letters
\[ \begin{align*}
\mu & \quad \text{dynamic viscosity, Pa sec} \\
\nu & \quad \text{kinematic viscosity, m}^2/\text{sec} \\
\rho & \quad \text{density, kg/m}^3 \\
\varepsilon & \quad \text{dissipation rate, m}^2/\text{sec} \\
k & \quad \text{turbulence kinetic energy, m}^2/\text{sec}
\end{align*} \]

REFERENCES


