Research Article

Bilal Yousaf, Mumtaz A. Qaisrani*, Muhammad Ijaz Khan, Muhammad Sana Ullah Sahar*, and Waqas Tahir

Numerical and experimental analysis of the cavitation and study of flow characteristics in ball valve

<https://doi.org/10.1515/nleng-2021-0044>

Received Aug 24, 2021; accepted Nov 29, 2021

Abstract: Cavitation in ball valve was numerically investigated using Computational Fluid Dynamics (CFD) modeling and then validated against results gained through experiments. The experiment was carried out in an assembly unit of the automotive industry to analyze flow patterns. The effect of bubbles on other thermophysical properties of the fluid was also examined using the multiphase $k-\epsilon$ viscous model in ANSYS FLUENT. The impact of changes in inlet pressure on vapor fraction was visualized through simulations and validated against experimental data, cavitation was calculated via cavitation index equation. It was observed that cavitation values ranged from 0.51 to 0.84 through computational fluid dynamics and from 0.46 to 0.80 in the experiment. Moreover, fluctuations in Turbulence Kinetic Energy (TKE) in the fluid through the boundary layers in the valve region, deformation in fluid particles in the form of Strain Rate (SR), and variations in the value of Wall Shear Stress (WSS) of the valve's internal walls were also studied through numerical simulations. The results show the pressure just before the valve drops and result in cavitation. Besides, turbulence kinetic energy, shear stress on the walls of a valve, strain rate, and fluid velocity were gradually increased at inlet pressure resulting erosion in the ball valve.

Keywords: Cavitation, ball valve, multiphase flow, vapor fraction, numerical modelling

Nomenclature

P_1	Inlet Pressure
P_o	Orifice (vena contracta) Pressure
P_2	Outlet Pressure
CFD	Computational Fluid Dynamics
C_F	Flow Coefficient
SR	Strain Rate
C_e	Cavitation Equation
TKE	Turbulence Kinetic Energy
P_v	Vapor Pressure
WSS	Wall Shear Stress
FVM	Finite Volume Method
R_c	Mass Transfer term with respect to Condensation
SIMPLE	Semi-implicit Method for Pressure Linked Equation Algorithm
Q or q	Flow Rate
RANS	Reynolds Averaged Navier Stokes Equation
R_e	Mass Transfer term concerning Evaporation
$V_{avg.}$	Volumetric Average Flow Velocity
V	Velocity
T_B	Bubble Surface Temperature
F_P	Geometric Factor
ρ_ℓ	Liquid Surface Tension Coefficient
G	Gravity w.r.t Water
ρ	Density
Re	Reynolds Number
α	Cavitation
ϵ	Turbulence Model
v	Vapor Phase
$\mathfrak{R}_B$	Bubble Radius
ρ_v	Vapor Density

*Corresponding Author: Mumtaz A. Qaisrani: Department of Mechanical Engineering, Khwaja Fareed University of Engineering and Information Technology, Rahim Yar Khan – 62400, Pakistan; Email: mumtazqaisrani@yahoo.com

*Corresponding Author: Muhammad Sana Ullah Sahar: Department of Mechanical Engineering, Khwaja Fareed University of Engineering and Information Technology, Rahim Yar Khan – 62400, Pakistan; Email: msanaullah@kfueit.edu.pk

Bilal Yousaf, Muhammad Ijaz Khan, Waqas Tahir: Department of Mechanical Engineering, Khwaja Fareed University of Engineering and Information Technology, Rahim Yar Khan – 62400, Pakistan

1 Introduction

Cavitation damage is a form of hyper-erosion that can destroy systems and reduces their life span, which can result in unacceptable process failures. The vapor bubbles created as a result of a pressure drop will grow and then collapse and the vapors return to liquid form. The implosion of vapor bubbles in the cavitation phenomenon inflicts damage in the form of small pits in the metal, which cumulatively wears away surfaces. Thus, inspections of cavitation are indispensable to detect defects at an early stage and maintain proper functioning of the systems.

Valves are mechanical devices intended to perform multiple functions such as controlling fluid pressure, mass, and direction of the fluid in a piping system. Cavitation occurs when fluid pressure suddenly drops below its vapor pressure, resulting in vibration or noise, which affects the ball valve's life span and hence must be controlled. By changing pressure and mass that flows through a system, cavitation can be avoided to some extent.

Figure 1 shows the cavitation process where the inlet the pressure inside the fluid channel is more than at the outlet. When the diameter of the channel changes, the pressure drops suddenly and the vaporization starts at this point. The cavitation starts when the pressure of the liquid decreases below its vapor pressure.

Figure 1: Pressure distribution of the fluid across the channel

As the pressure of the liquid increases again, the bubbles collapse transforms to liquid phase. A typical ball control valve is shown in Figure 2. It consists of a ball, seat rings, main body of valve, stem, packing gland, valve end, and handle lever.

Considering this, Yuzawa *et al.* [1] used a contoured plug valve manufactured from three different materials, stainless steel, stellite, and brass, and used water as the working fluid to access cavitation along with material loss due to erosion which affects the overall performance. Nie

Figure 2: Structural model of ball control valve

et al. [2], using water as a working fluid, calculated cavitation characteristics in a two-stage throttle valve by using the CFD technique and proved that a high amount of back-pressure is significant for the reduction of cavitation.

Bernad *et al.* [3] studied the RNG $k-\epsilon$ turbulence model and used a poppet valve along with water as a working fluid and found that increase in inlet pressure increases cavitation. Liu *et al.* [4] used a butterfly valve and water including particles as a working fluid and predicted cavitation erosion using ANSYS FLUENT. Jin *et al.* [5], Shirazi *et al.* [6], and Tabrizi *et al.* [7] used a globe and ball valves and simulated the cavitation process in ANSYS FLUENT, leakages in the piping systems was due to cavitation that causes severe damages and also reduces the life expectancy of the control valves. Chern *et al.* [8] used a ball valve and experimentally studied the cavitation process. Similarly, Nzombo D. *et al.* [9], using butterfly and ball valves, simulated the cavitation process, and design parameters were optimized using algorithms, the impact of cavitation on design parameters was investigated.

Cui *et al.* [10] simulated the cavitation using ANSYS FLUENT and validated the findings experimentally during opening and closing of the ball valve, the performance of the control valve was affected. A.M. Abdulaziz [11] used a venturi and experimentally investigated the cavitation, and then validated the results of vapor volume fraction through an image analysis technique. In another study [12], researchers experimentally and numerically investigated that cavitation in poppet valve. They also performed simulations of the mixture model using the $k-\epsilon$ turbulence model. Using this model, they predicted cavitation near the outlet (after the valve region), as the intensity of cavitation was significantly affected by the recirculation zones in downstream regions. In another series of experiments Mancuso *et al.* [13, 14] also investigate hydrodynamic cavitation in wastewater treatment plants, and effect of different factors on cavitation effectiveness were also evaluated. Later [15], a jet reactor was used to produce cavitation, and the effect

of that reactor on solubilization, and their effects on different parameters (like temperature, inlet pressure etc.) were evaluated.

The literature review reveals that cavitation in ball valves using the Computational Fluid Dynamics (CFD) techniques remained understudied. The current research endeavors to fill this research gap. In this manuscript, cavitation was investigated using ANSYS FLUENT software, and the simulations were performed using the CFD method that relies upon the Finite Volume Method (FVM). Reynolds Averaged Navier Stokes (RANS) equation was solved using the $k-\epsilon$ standard mixture multiphase model and the vapor volume fraction (cavitation) and pressure distribution inside a ball valve with the no-slip condition. The result of cavitation was compared with experimental data to validate the results.

The proposed CFD strategy was used to find the physical appearance of cavitation that cannot be investigated experimentally. This paper problem is related to water supply lines from an automotive assembly unit, where ball valves are used to control the fluid flow. Ball valves were used to control the fluid pressure. To visualize cavitating flow and the intensity of cavitation the Schnerr-Sauer cavitation model was employed. The results from the numerical simulation were compared with the experimental data.

The experiment was carried out to validate the numerical simulation and the flow coefficient equation was used to measure the quantity of the flowing liquid passing through the channel per unit time, and the cavitation equation was used to predict the cavitation. The numerical simulation was performed with the steady-state conditions in ANSYS FLUENT, and an experiment was also carried out in steady-state conditions. From simulations, the effect of inlet pressure on cavitation intensity was calculated, and the change in vapor volume fraction, the flow coefficient, wall shear stress, strain rate, flow rate, and turbulence kinetic energy were also examined.

2 Numerical simulations

The mathematical modeling of the cavitation, based on the finite volume method presented in section 2.2, was used to simulate the process, and a multiphase mixture model was employed to calculate vapor volume fraction.

The simulations were performed in steady-state condition. At the inlet of the valve, pressure varies from 1.0×10^5 to 6.0×10^5 Pa whereas the outlet pressure *i.e.*, 9.5×10^4 Pa, was kept constant during the whole simulation. The selected inlet pressures were based on required pressure in the au-

tomotive industry, where the experiment was performed. Similar values of inlet pressure were analyzed experimentally. For cavitation, the Schnerr Sauer cavitation model in ANSYS FLUENT was used, in which the Reynolds Averaged Navier Stokes (RANS) equation was solved with the standard $k-\epsilon$ multiphase mixture model with standard wall functions. The swirling flow conditions were best measure with $k-\epsilon$ turbulence model. The turbulent flow conditions within the flow were calculated by using this model rather than $k-\omega$ model. Because $k-\omega$ model is only beneficial for measuring values near the wall (in shear flow conditions). The fluid mixture model consisted of two fluids, *i.e.*, water and water vapor. The density for water and water vapors was set up as 1×10^3 kg/m³ and 0.01 kg/m³, respectively, with the diameter of the water vapor taken as 1.0×10^{-5} . The vapor pressure (P_v) and bubble density were set to 3×10^3 Pa and 1×10^{13} , respectively, for cavitation inception. The supersonic gauge pressure was set to 4.0×10^4 Pa to initiate the pressure at the inlet of the ball valve, and the cavitation was investigated. For pressure, the coupling scheme was used, where pressure and convergence criteria for momentum and turbulence kinetic energy were set to second-order upwind, while dissipation rate to first-order upwind. The QUICK scheme was used for the vapor volume fraction, and the relaxation factor for higher-order terms was set to 0.95. Details of the ANSYS FLUENT mixture multiphase model was mentioned in Table 1.

As shown in Table 1, the discretization scheme for momentum and turbulence was set to second-order upwind. The semi-implicit method for pressure-linked equations (SIMPLE) algorithm is used in the solver, considering its

Table 1: ANSYS FLUENT solver details

Equations	Mixture model
Solver	SIMPLE
Turbulence model	$k-\epsilon$
Wall treatment	No-slip
Cavitation model	Schnerr and Sauer
Discretization of the pressure equation	PRESTO
Discretization of vapor fraction	QUICK
Discretization of momentum and turbulence	Second-order upwind
Under relaxation of vapor fraction	0.001
Inlet pressures	1.0×10^5 to 6.0×10^5 Pa
Outlet pressures	9.5×10^4 Pa
Density of liquid	1000 kg/m ³
Fluid used	Water, Vapor

Table 2: Boundary conditions & comparison of Cavitation values (experiment and simulation data)

Inlet Pressure (Pa)	Outlet Pressure (Pa)	Cavitation (Experimental values)	Cavitation (Simulation values)
1.0×10^5	9.5×10^4 Pa	0.46	0.51
2.0×10^5	9.5×10^4 Pa	0.53	0.65
3.0×10^5	9.5×10^4 Pa	0.69	0.70
4.0×10^5	9.5×10^4 Pa	0.76	0.73
5.0×10^5	9.5×10^4 Pa	0.79	0.79
6.0×10^5	9.5×10^4 Pa	0.80	0.84

efficiency in such simulations [16, 17]. The cavitation Schnerr and Sauer model was used to visualize the dynamics of bubbles within the channel. The intensity of cavitation, length of cavitation, and flow loss were also calculated. The results obtained from the cavitation model were accurate and within an acceptable range of experimental results. The summary of boundary conditions and cavitation values from experiment and numerical simulations are mentioned in Table 2.

2.1 Grid independence test

A grid independence test was carried out using the Richardson extrapolation method [18], with a refinement ratio higher than 1.3. The mesh independence details concerning volumetric average flow velocity (V_{avg}) are given in Table 3.

Table 3: Mesh independence details

Case	No. of Elements	Volumetric average flow velocity (V_{avg})
1	90614	24.52
2	128673	24.53
3	258634	24.55
4	331546	24.55
5	410458	24.55

The data presented in Table 3 elucidates the fact that a good quality mesh was achieved with an orthogonal value of 0.974, skewness of 0.24, and an error of less than 1% (case 3). As shown in Table 3, mesh elements increased to 258634, the volumetric average flow velocity (V_{avg}) was 24.55, which remains the same on any further increase in the mesh elements. Hence 258634 mesh elements were con-

sidered for the current case. In addition, mesh quality near the vena contracta region was focused on analyzing the best results of cavitation.

2.2 Governing equations for CFD modeling

The Rayleigh-Plesset equation was used to understand the dynamics of a bubble's inception in cavitation. In a fluid channel, the air enters into the channel and forms bubbles, resulting in increased cavitation. Figure 3 shows a bubble having radius, temperature, and pressure 'R', P_B , and T_B , respectively, whereas the pressure and temperature of the whole fluid-channel are denoted by P_∞ and T_∞ , respectively.

Figure 3: Schematic of a spherical bubble in an infinite liquid

The Rayleigh-Plesset equation [19] is as follows:

$$\frac{p_B(t) - p_\infty(t)}{\rho_L} = R \frac{d^2 R}{dt^2} + \frac{3}{2} \left(\frac{dR}{dt} \right)^2 + \frac{4\gamma_L}{R} \frac{dR}{dt} + \frac{2S}{\rho_L R}. \quad (1)$$

In a cavitation model, the liquid-vapor mass transfer is directed by the vapor transport equation [20], where α is cavitation (vapor volume fraction) v is the vapor phase, ρ_v is the vapor density, V_v is the vapor phase density, R_e , R_c and are mass transfer terms.

$$\frac{\partial}{\partial t}(\alpha \rho_v) + \nabla \cdot (\alpha \rho_v V_v) = R_e - R_c. \quad (2)$$

In industrial settings, the bubble dynamics equation for the formation and collapse of the bubbles with no-slip velocity condition can be derived from Rayleigh-Plesset equation, i.e.,

$$\mathfrak{R}_B \frac{D^2 \mathfrak{R}_B}{Dt^2} + \frac{3}{2} \left(\frac{D \mathfrak{R}_B}{Dt} \right)^2 = \left(\frac{P_B - P}{\rho_\ell} \right) - 4 \frac{\nu_\ell}{\mathfrak{R}_B} - \frac{2S}{\rho_\ell \mathfrak{R}_B}. \quad (3)$$

Where, $\mathfrak{R}_B$ is the bubble radius, ρ_ℓ is the liquid surface tension coefficient, P is the pressure, and P_B is the bubble

surface pressure. The Shneur and Sauer model was implemented in the current study to determine the cavitation (vapor volume fraction). Mathematically, the cavitation (vapor volume fraction) can be represented as:

$$\frac{\partial}{\partial t}(\alpha \rho_v) + \nabla \cdot (\alpha \rho_v V_v) = \frac{\rho_v \rho_l}{\rho} \frac{D\alpha}{Dt} \quad (4)$$

and

$$R = \frac{\rho_v \rho_l}{\rho} \frac{D\alpha}{Dt}. \quad (5)$$

Here, R is the mass transfer term. In the above equation, α is known as vapor volume fraction, and the bubble quantity can be found during cavitation using the following formula

$$\alpha = \frac{n_b \cdot \frac{4}{3} \pi \mathfrak{R}_B^3}{1 + n_b \cdot \frac{4}{3} \pi \mathfrak{R}_B^3}. \quad (6)$$

Like other approaches, Shnerr and Sauer cavitation model aids to calculate the rate of mass transfer using the following mathematical expressions:

$$R = \frac{\rho_v \rho_l}{\rho} \alpha (1 - \alpha) \frac{3}{\mathfrak{R}_B} \sqrt{\frac{2}{3} \frac{(P_v - P)}{\rho_l}}. \quad (7)$$

Here $\mathfrak{R}_B$ was used to measure the radius of the bubble. $\frac{\rho_v \rho_l}{\rho} \alpha (1 - \alpha)$ varies directly to the Shnerr Sauer model, and this function approaches zero at $\alpha = 0$ and $\alpha = 1$, and its maximum value in between. The final form of the Shnerr Sauer model can be written as:

$$R_e = \frac{\rho_v \rho_l}{\rho} \alpha (1 - \alpha) \frac{3}{\mathfrak{R}_B} \sqrt{\frac{2}{3} \frac{(P_v - P)}{\rho_l}}, \quad \text{when } P_v \geq P \quad (8)$$

and,

$$R_c = \frac{\rho_v \rho_l}{\rho} \alpha (1 - \alpha) \frac{3}{\mathfrak{R}_B} \sqrt{\frac{2}{3} \frac{(P - P_v)}{\rho_l}}, \quad \text{when } P_v \leq P \quad (9)$$

Here R_e , R_c are the mass transfer terms representing the formation and collapse of vapors, respectively.

3 Experimental setup and cavitation test

The ball valve investigated in this study was from a water supply line in the automotive industry. The flow meter

was attached to the line before the ball valve. The pressure gauge was also attached near the inlet of the valve and another pressure gauge was attached in the downstream region as shown in Figure 4.

The erosion in the ball valve is a major problem that appeared in the automotive industry, and the erosion damage is a result of cavitation within the flow channel, which in turns damages the whole assembly at times. The section view of the ball valve designed in Solidworks to understand the changes in flow pattern is shown in Figure 5(a) whereas Figure 5(b) shows an eroded ball valve on which an experiment was performed.

From Figure 5(b) it is clearly shown that the cavitation has occurred, during an experiment when the ball valve was partially open. The erosion on the inner layer of the ball is seen in Figure 6(a). The performance of the ball valve is also affected due to erosion.

In Figures 6(a) and 6(b) it is shown that the valve end and the ball of the ball valve are affected due to cavitation. The erosion layer can be seen on the inside section of ball valve, where cavitation occurred. When the pressure drops below the saturated vapor pressure of the liquid, then cavitation is incepted. But as the pressure rise above the vapor pressure, the bubbles collapse, and force is exerted in an outward direction. The internal layer of the ball valve is damaged.

The Figure 7 shows the conceptual diagram representing the bend in the flowing channel that describes the partially open condition of the ball valve in such a way that when the high-pressure fluid reaches the bend region, fluid flow is laminar and has maximum kinetic energy. But when it strikes the orifice or sudden blockage within the channel, pressure drops, and when local pressure drops below the vapor pressure of the liquid, the cavitation is incepted. The velocity and intensity of turbulence increase during this process of formation and collapse of bubbles (cavitation).

From Figure 7, it can be seen that the fluid entering the channel with a certain inlet pressure, and flow domain remains laminar. However, as the diameter of the channel changes, a sudden pressure drop occurs and maximum energy is dissipated in this region. The flow velocity increases and the laminar flow changes to turbulent. After passing through the vena contracta, when the fluid strikes

Figure 4: Schematic diagram of the experiment performed

Figure 5: (a) – Cross-sectional view of the ball valve. (b) – Erosion shown in the partially open ball valve

Figure 6: Damaged parts of the ball valve due to cavitation. (a) – Internal view of the eroded ball (b) – eroded valve end

Figure 7: Conceptual design of flowing fluid through ball valve

Figure 8: (a) – Pressure distribution (pressure drop at vena contracta) shown at different inlet pressures. (b) – Vapor fraction (cavitation) was represented at different inlet pressures ranging 1–6 bars

Figure 9: Contour of molecular viscosity at different inlet pressures (1–6 bars)

the channel outside the ball valve at that point the swirls are produced, as shown in Figure 10(a). Figure 7 also shows that the cavitation is incepted near the edges and the critical cavitation zone is the area where maximum bubbles or clusters of bubbles are formed. When the fluid leaves the ball valve some bubbles are also present. The formation of bubbles is due to sudden pressure drop below the vapor pressure of the liquid and again when pressure increases with no time, the bubbles collapse *i.e.*, cavitation occurs.

4 Results and discussions

4.1 Numerical results

The cavitation was visualized and analyzed through numerical techniques. The strain in the particles of the fluid during and after cavitation was also studied. The effect of force on the internal walls of the ball valve due to bubble collapse was also analyzed. Figure 8(a) shows pressure distribution contours for the fluid as it flows through the pipe encompassing the ball valve. The cavitation can be observed at that narrow place, *i.e.*, vena-contracta, as the bubbles are form and collapse at the corner edges of the ball valve which was partially opened. The velocity of the fluid increases because the diameter of the channel changes suddenly, the pressure just before the valve decreases, and a recirculation vortex appeared near the edges of the ball valve. The color spectrum at the corner of the edges showed that there is a sudden pressure drop, which resulted in cavitation at those edges and in vena contracta.

Figure 8(b) shows that cavitation initiated at low pressure and as the pressure increases the cluster of bubbles increases *i.e.*, a change in vapor volume fraction (cavitation) during the partial opening of the ball valve at different inlet pressures can be observed. It can be seen that the vapor volume fraction (cavitation) gets higher at the edges of the valves. Because of sudden pressure drop and high velocity, the pressure drops below its vapor pressure, and the vapor fraction appears in the fluid channel. Theoretically, [6] vapor volume fraction appears when the pressure drops below the saturated vapor pressure. The length of

Figure 10: (a) – Change in velocity magnitude at inlet pressures from 1–6 bars. (b) – Turbulent kinetic energy change was illustrated at pressures 1–6 bars in ball valve

cavitation also increased as the inlet pressure increased from 1 to 6 bars in a partially opened ball valve, as shown in Figure 8(b). The erosion also occurs in these regions, as clearly shown in Figure 8(b).

From Figure 8(a), it can be clearly seen the pressure changes in the system and production of vortices are also observed at vena contracta region. The vortices at the inlet of the ball valve are smaller than outlet of the valve. The inception of cavitation was seen at the brink of vortex. When cavitation happens, the abnormal sound produced in the system.

Figure 9 shows molecular viscosity contour at different inlet pressures, and it can be observed that the formation of the bubbles. The erosion takes place on the edges of the valve as the bubbles collapse. Fluid viscosity in the whole system remains the same but changes with a sudden drop in pressure and liquid changes its phase for a while, the changes in molecular viscosity is due to the inception of cavitation in the flow. If the flowing fluid's viscosity is high, then the trailing force reaches its highest point, and the pressure inside the vapors was not so high that is why the vapors did not develop and collapse suddenly, and finally, the cavitation strength reduces with increasing viscosity. These findings also verify that the growth and collapse of vapors decrease with an increase in the viscosity of the flowing fluid [21]. From Figure 9, it is clear that at low inlet pressure the small number of liquid changes to bubbles during cavitation. But as the inlet pressure increases, the cavitation region increases.

Figure 10(a) illustrates that, as the pressure drops down at vena contracta, the velocity increases. The recirculation

area behind the valve increases as the velocity increases near the vena contracta, and with the increase in the inlet pressure, the fluid particles have more kinetic energy, and thus more energy is dissipated near the partially open valve. It is concluded that the recirculation length increases with the increase in the inlet pressure in a partially open valve. In other words, the system must spend more energy to preserve the same volumetric flow rate at different inlet pressures in a partially open valve.

Figure 10(b) shows that when the inlet pressure increased from 1 to 3 bars, the kinetic energy of the fluid particles shows a little turbulence, because the pressure drop was also low at vena contracta and a little change in velocity occurred as shown in Figure 10(a). But with the increase in pressure from 3 to 6 bars, the turbulence increases because at high pressure the fluid particles have more kinetic energy and more energy is dissipated at high pressure. When at high pressure the fluid changes its path due to an orifice, the changes in velocity gradient were also high as observed and shown in Figure 10(a). For the same reason the streamlined fluid goes turbulent, and the intensity of turbulence goes up with increasing pressure.

4.2 Experimental results

To validate the numerical simulation, an experiment was performed in an automotive assembly unit. Experimentally the value of the valve can be calculated using two coefficients *i.e.*, the cavitation equation and the flow coefficient. Kirik and Driskell [22] also used the same for cavitation. The

flow coefficient in piping systems is measured by flow meter as shown in Figure 4. The flow coefficient is the measure of fluid flowing through the valve, which is denoted as:

$$C_F = \frac{q}{0.865 F_P} \sqrt{\frac{G}{\Delta P}} \quad (10)$$

In Eq. (10), G is the gravity concerning water and F_P is the geometric factor. Both values are assumed to be unity in this study. To calculate the cavitation within the flow, a cavitation equation was used, which was defined as:

$$C_e = \frac{\Delta P}{P_{in} - P_v} \quad (11)$$

Where P_{in} is the inlet pressure, P_v is the saturated vapor pressure of the flowing liquid. ΔP is the pressure gradient between the inlet and the outlet pressure.

In the current experiment, the value of vapor fraction (cavitation) is calculated from Eq. (11), ranging from 0.46 to 0.80, and verification of results from numerical simulations whose values are found to be in the range of 0.51 to 0.84, as shown graphically in Figure 11. From Figure 11, it is clear that the cavitation increases with the increase in the inlet pressure. This is because, at high pressure, fluid particles have more kinetic energy than at low pressure. Moreover, vapor inception at high pressure is more than at low-pressure value in the partial open ball valve.

Figure 11: Validation for cavitation by numerical simulations against experimental data

In the laminar flow the fluid having of Reynolds Number (Re) was ≥ 2300 , and when the pressure suddenly drops within the channel the turbulency produces within the flow of the fluid due to high velocity and the value of Reynolds Number (Re) was ≤ 4000 . The Figure 14, shows the change in Reynolds Number (Re) at the vena contracta region, where the fluid flow changes to turbulent. As can be observed the values of inlet pressures plotted against the Reynolds Number (Re) increases because it is high initially at inlet pressure the velocity of the fluid was also high, and when the

pressure drops below saturated vapor pressure the value of Reynolds number changes to an increased value than at low inlet pressure.

As the fluid move with some inlet pressure through a channel, it has certain amount of kinetic energy, however, due to a sudden drop in local pressure then, because of the orifice, the flow of the liquid changes to turbulent, and their Reynolds number changed. As shown in Figure 12, at low inlet pressure because the velocity of the fluid has less value at 1 bar than at 6 bar, and less turbulence is produced at 1 bar pressure. That's why the value of Reynolds number is also low at 1 bar inlet pressure than at 6 bars inlet pressure.

Figure 12: Graph between Reynolds Number and turbulence kinetic energy at varying inlet pressures

Figure 12 also illustrates the change in turbulence kinetic energy with an increase in the inlet pressure. It also shows that by increasing the inlet pressure, the turbulence in the fluid increases at the valve region, and laminar flow changes to turbulent; thereby, the velocity of the flowing fluid increases suddenly due to sudden pressure drop. Due to a continuous increase in inlet pressure, the turbulency in the fluid particles increases at vena contracta, and the value of the Reynolds number also increases. Increased pressure shows higher cavitation because 6 bar than at lower pressure. For the same reason the turbulence effect is frequently visualized in fluids having low viscosity.

Figure 13 demonstrates a relation between Pressure (P), Wall Shear Stress (WSS), and Strain Rate (SR). As the inlet pressure in the partial open ball valve increases, the deformation in the fluid properties occurs because the fluid particles face some sudden blockage which also results in the formation of vortices. Figure 13 also clearly demonstrates that a change in the inlet pressure from 1 to 6 bars imparts shear stress to walls of the ball valve, which interacts with the high-pressure fluid. When the pressure drops suddenly due to an orifice in the system the turbulence produces due

Figure 13: Graph representing the strain in liquid particles in cavitating flow and also change in shear stress produced due to cavitation for inlet pressures

to high velocity of the fluid stream. Comparatively more turbulence is produced at increased inlet pressure rather than at low pressure, and due to high turbulence, the secondary flow was induced due to high turbulence at vena-contracta. When the pressure suddenly drops due to a partially opened valve, deformation in the solid particles also occurs due to low pressure and high-velocity fluid in narrow regions in the system. The force applied on internal walls of ball valve due to bubble collapse, and if that applied force is more than that of external pressure then the valve damages or sometimes the whole system collapses. Wall shear stress curve clearly shows that at low pressure the shear force is also minimum as compared to high pressure. Hence shear stress is produced within the system the increases by increasing the inlet pressure.

Figure 14: Graph comparing the values of flow rate flowing through the partial open ball valve by numerical simulation and experiment

The flow rate of the fluid was examined both numerically and experimentally. From Figure 14, it is seen that by changing inlet pressure from 1 to 6 bars, the rate of fluid flow changed respectively. In Figure 14, the numerical values of changing flow rate are shown with the circular symbol and compared those values from experimental data (as shown

with the triangular symbol) which shows some standard deviation in values, but overall results are approximately similar. At higher inlet pressure of fluid flowing through the partially opened ball valve, a higher flow rate was observed. The parity plot between the fluid flowrate between the numerical simulation and the experimental data shows a difference of not more than 3% initially which further reduces as the pressure of the inlet increases.

5 Conclusions

Cavitation was analyzed in a ball valve under the Schnerr Sauer cavitation model using ANSYS FLUENT. To verify the numerical simulation results an experiment was performed. The inlet pressure of the valve varies from 1.0×10^5 to 6.0×10^5 Pa, the outlet pressure of 9.5×10^4 Pa was kept constant. Similar pressure was applied in an experiment to justify the numerical results. The result shows that the pressure drops down at vena contracta, the cavitation was observed in the fluid stream. The $k-\epsilon$ viscous model predicted the turbulence in the fluid and, Schnerr and Sauer model was used to calculate the cavitation numerically. Cavitation values were approximated in numerical simulations ranged from 0.51 to 0.84 and when the values measured experimentally were in the range of 0.46 to 0.80, respectively, at different inlet pressures ranging from 1.0×10^5 to 6.0×10^5 Pa in a partial open ball valve. The thermophysical properties which were affected due to cavitation were also studied. It was concluded that turbulence kinetic energy increased during the cavitation. The wall shear stress occurred outwards due to the collapse of bubbles. The flow rate was also investigated both numerically and experimentally. The approximately similar results were shown. The impact of cavitation on Reynolds number was also investigated. The changes in the flow pattern were also studied because the deformation in the molecules of the fluid occurred during cavitation which increased by increasing the inlet pressure in the partial open ball valve. This study would also be helpful in automotive industry applications where high-pressure fluid was flowing through channels. In that case, noise and vibration effects were also observed. To control these phenomena of vibration and noise, different experimental techniques were applied but those techniques sometimes harm the other equipment also. For that purpose, it was necessary to overcome such problems at design stages using numerical techniques to resolve the erosion and cavitation problems in the ball valve.

Acknowledgement: The author(s) received no specific funding for this work.

Author contributions: All authors have accepted responsibility for the entire content of this manuscript and approved its submission.

Conflict of interest: The authors state no conflict of interest.

Data availability: The data that support the findings of this study are available from the corresponding author upon reasonable request.

References

- [1] Yuzawa S, Okutsu R, Hashizume T, Outa E. Cavitation erosion features in industrial control valves at an inlet pressure of 20 MPa. *JSME Int J Ser B Fluids Therm Eng.* 1998 Nov;41(4):1105–13.
- [2] Nie S, Huang G, Li Y, Yang Y, Zhu Y. Research on low cavitation in water hydraulic two-stage throttle poppet valve. *Proc Inst Mech Eng, E J Process Mech Eng.* 2006 Aug;220(3):167–79.
- [3] Bernad S, Susan-Resiga R, Muntean S, Anton I. Cavitation phenomena in hydraulic valves. *Numerical modelling. P Romanian Acad A.* 2007 Feb;8(2).
- [4] Liu B, Zhao J, Qian J. Numerical analysis of cavitation erosion and particle erosion in butterfly valve. *Eng Fail Anal.* 2017 Oct;80:312–24.
- [5] Jin ZJ, Gao ZX, Qian JY, Wu Z, Sundén B. A parametric study of hydrodynamic cavitation inside globe valves. *J Fluids Eng.* 2018 Mar;140(3):031208.
- [6] Shirazi NT, Azizyan GR, Akbari GH. CFD analysis of the ball valve performance in presence of cavitation. *Life Sci J.* 2012;9(4):1460–7.
- [7] Tabrizi AS, Asadi M, Xie G, Lorenzini G, Biserni C. Computational fluid-dynamics-based analysis of a ball valve performance in the presence of cavitation. *J Eng Thermophys.* 2014 Jan;23(1):27–38.
- [8] Chern MJ, Wang CC, Ma CH. Performance test and flow visualization of ball valve. *Exp Therm Fluid Sci.* 2007 May;31(6):505–12.
- [9] Nzombo D. A numerical investigation of cavitation in valves and techno-economic analysis of Pinewood solvent liquefaction [dissertation] Ames (IA): Iowa State University. <https://doi.org/10.31274/etd-180810-5210>.
- [10] Cui B, Lin Z, Zhu Z, Wang H, Ma G. Influence of opening and closing process of ball valve on external performance and internal flow characteristics. *Exp Therm Fluid Sci.* 2017 Jan;80:193–202.
- [11] Abdulaziz AM. Performance and image analysis of a cavitating process in a small type venturi. *Exp Therm Fluid Sci.* 2014 Feb;53:40–8.
- [12] Hong G, Xin F, Hua-yong Y, Tetsuhiro T. Numerical investigation of cavitating flow behind the cone of a poppet valve in water hydraulic system. *J Zhejiang Univ Sci.* 2002 Sep;3(4):395–400.
- [13] Mancuso G, Langone M, Andreottola G. A critical review of the current technologies in wastewater treatment plants by using hydrodynamic cavitation process: principles and applications. *J Environ Health Sci Eng.* 2020 Jan;18(1):311–33.
- [14] Mancuso G, Langone M, Laezza M, Andreottola G. Decolourization of Rhodamine B: A swirling jet-induced cavitation combined with NaOCl. *Ultrason Sonochem.* 2016 Sep;32:18–30.
- [15] Mancuso G. Experimental and numerical investigation on performance of a swirling jet reactor. *Ultrason Sonochem.* 2018 Dec;49:241–8.
- [16] Versteeg HK, Malalasekera W. An introduction to computational fluid dynamics: the finite volume method. 2nd ed. Pearson Education; 2007.
- [17] Tu J, Yeoh GH, Liu Ch. *Computational Fluid Dynamics, A Practical Approach.* Elsevier; 2019.
- [18] Roache PJ. Perspective: a method for uniform reporting of grid refinement studies *J Fluids Eng.* 1994;116(3):405–413. <https://doi.org/10.1115/1.2910291>.
- [19] Brennen Christopher E. *Cavitation and Bubble Dynamics.* United Kingdom: Oxford University Press; 1995.
- [20] Liu X, Wu Z, Li B, Zhao J, He J, Li W, et al. Influence of inlet pressure on cavitation characteristics in regulating valve. *Eng Appl Comput Fluid Mech.* 2020 Jan;14(1):299–310.
- [21] Poritsky H. The growth or collapse of a spherical bubble or cavity in a viscous fluid. In *Proceedings of the First US National Congress on Applied Mechanics*; 1951 Jun 11–16; Chicago, USA. ASME, 1952 p. 813–821.
- [22] Kirik MJ, Driskell LR. *Flow Manual for Quarter-Turn Valves.* Rockwell International. Flow Control Division; 1986.