Asian Journal of Scientific Research

ISSN 1992-1454

Asian Journal of Scientific Research 6 (3): 581-588, 2013
ISSN 1992-1454 / DOI: 10.3923/ajsr.2013.581.588

Numerical Investigation of Flow in Hydraulic Valves with Different Head Shapes

Javad Taghinia-Seyedjalali and Timo Siikonen
Department of Applied Mechanics, School of Engineering, Aalto University, 00076, Espoo, Finland

Corresponding Author: Javad Taghinia-Seyedjalali, Department of Applied Mechanics, School of Engineering, Aalto University, 00076, Espoo, Finland

ABSTRACT

Valves are one of the main components of fluid control devices in applications such as oil and gas machinery. They play an important role in the efficiency of these systems in terms of energy and performance. Therefore, understanding the flow characteristics and behavior in valves is vitally important in the design and optimization of such systems. A Computational Fluid Dynamics (CFD) simulation is performed using an in-house code for ball and conical head poppet valves. The flow is turbulent, incompressible and unsteady. In order to obtain good and converged results, the RNG-k-ε turbulence model is applied for this purpose. Simulations apply a velocity inlet and pressure outlet. The effect of different gap sizes and the shape of the valve are investigated. The results are compared with each other and with available experimental data. They also show good agreement with the works found in the literature. It was concluded that the shape of the valve has a dominant effect on flow behavior, while recirculation of the flow and a high-velocity region in the gap are the main characteristics of the flow in valve systems.

Key words: CFD, Hydraulic valves, RNG k-ε turbulence model

INTRODUCTION

Over the last two decades much attention has been given to the use of CFD as a method to investigate flow behavior in various systems. With the development of powerful computers and computational tools, CFD has become popular in different fields of fluid dynamics such as fluid power and control systems. Valves are one of the main components of the fluid control systems and they play an important role in the efficiency of these systems. Understanding the flow characteristics and behavior in valves is therefore vitally importance in the design and optimization of such systems (Mokhtarzadeh-Dehghan *et al*., 1997). There are many valves with different shapes and applications. In this paper, a poppet valve, one of the most common in control systems, is considered. The poppet valve is a seating type valve in which the moving element usually has a spherical or conical shape that moves in a direction parallel to the axis of a seat.

The present study uses numerical simulation and a RNG k-ε model in order to gain a better understanding of the fluid behavior in spherical and conical poppet valves. A number of previous studies have applied numerical methods to simulate fluid flow in valves. Pounty *et al*. (1989) carried out a simulation of a water flow in a servo-valve orifice using the k-ε turbulence model for different valve gaps. Vaughan *et al*. (1992) simulated the steady turbulent flow through a poppet valve. Their results showed good agreement with experimental data. Other researchers, like Leino *et al*. (2003) studied the pressure and velocity distributions in a water hydraulic valve. The effect of various types of grid systems and different turbulence models have been investigated.

Tsukiji *et al*. (1995) studied the flow pattern in a poppet valve by flow visualization and numerical simulation. Sorensen (1999) used experimental and CFD analyses of internal flow in a conical seat valve for different Reynolds numbers. Ueno *et al*. (1994) conducted an experimental investigation of an oil flow in a pressure control valve under a non-cavitating condition and concluded that the cavitation is the main source of noise in valves and that the valve configuration is one of the main factors in producing noise. In this study, the flow characteristics of spherical and conical head poppet valve with a two-dimensional axisymmetric configuration are investigated. Two results for the flow behavior are then compared to determine the effect of valve's head shape in order to improve its performance.

MATHEMATICAL MODELING

To simplify the problem, a 2D axisymmetric geometry is proposed in which the fluid is assumed as incompressible and fully turbulent. The RNG k-ε model is derived on the basis of Navier-Stokes equations (Yakhot and Orszag, 1986).

The momentum conservation equations are given by:

$$\frac{\partial U_i}{\partial t}+U_i\frac{\partial U}{\partial x_j}=-\frac{1}{\rho}\frac{\partial P}{\partial x_i}+\vartheta\frac{\partial^2 U_i}{\partial^2 x_j}+\frac{1}{\rho}\frac{\partial\left(-\rho u'_i u'_j\right)}{\partial x_j} \tag{1}$$

Transport equations for this model are similar to the standard k-ε model:

$$\rho\frac{Dk}{Dt}=\frac{\partial}{\partial x_j}\left(\alpha_k\rho C_\mu\frac{k^2}{\in}\frac{\partial k}{\partial x_i}\right)-\frac{\overline{\rho u'_i u'_j}\,(\partial U_j)}{\partial x_j}-\rho\in \tag{2}$$

For the dissipation term:

$$\frac{\rho\frac{D\in}{Dt}=\frac{\partial}{\partial x_j}\left(\alpha_\in\rho C_\mu\frac{k^2}{\in}\frac{\partial k}{\partial x_i}\right)-C1_\in\frac{\in}{k}\overline{\rho u'_i u'_j}\,(\partial U_j)}{\partial x_j}-C_2\in\rho\frac{\in^2}{k}-R \tag{3}$$

where, k is the turbulent kinetic energy, ε is the dissipation rate, $C_{1\varepsilon}$ = 1.42 and $C_{2\varepsilon}$ = 1.68 are constants, α_k and α_ε are the inverse effective Prandtl numbers for k and ε. R is a source term:

$$R=\frac{C_\mu\rho\left(1-\frac{\eta}{\eta_0}\right)}{1+\beta\eta^2}\frac{\in^2}{k} \tag{4}$$

where, η is a dimensionless parameter, η = 0.0845, η_0 = 4.38 and β = 0.012. The turbulent viscosity can be calculated as:

$$\mu_t=\rho C_\mu\frac{k^2}{\in} \tag{5}$$

GEOMETRY AND GRID SYSTEM

Because of the complexity of the geometry, the problem is simplified to a 2D axisymmetric case. The main components of this system are a valve face and a valve seat (Fig. 1). The area of interest in this study is where the valve face and seat meet each other, which controls the flow rates inside the rectangular domain. A structured grid is used for meshing the model. A few test runs were made with different grid densities to determine the number of cells needed for the calculations. The gap between the valve surface and seat was set to 0.4 and 0.2 mm for case 1 and 2, respectively.

BOUNDARY CONDITION

The inlet is set to a velocity of 5 m sec^{-1}. No-slip wall conditions are imposed on solid boundaries. The walls are assumed to be adiabatic and, therefore, no heat transfer occurs between the fluid zone and solid zone. At the outlet the pressure is specified as 1 bar (Fig. 2). At the symmetry axis the gradients of all the variables are set to zero. The fluid is water with a viscosity of = 0.001003 kg m sec^{-1}.

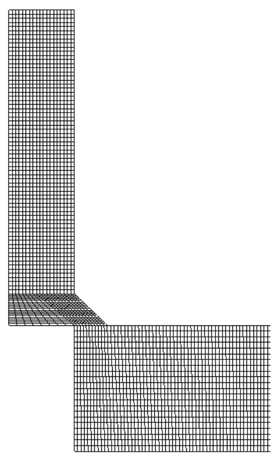

Fig. 1: Grid system for the valve

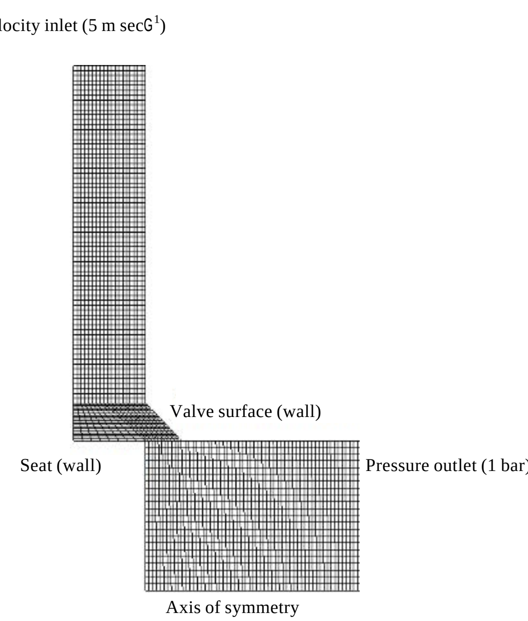

Fig. 2: Boundary conditions for the model

RESULTS AND DISCUSSION

Here, a test model is compared with the works of Mokhtarzadeh-Dehghan *et al.* (1997) for the purpose of validation. Two types of valves are used to investigate the effect of different gap spaces and valve shapes of on the flow behavior in the computational domain.

Validation: A model is developed according to a geometry proposed in the works of Mokhtarzadeh-Dehghan *et al.* (1997). In this model the inlet experiences a 200 bar pressure and the gap between the surface of the valve and seat is 1 mm. The outlet is a stress-free condition outflow boundary that makes the pressure close to zero. The test model showed good agreement for the velocity.

In both cases the maximum velocity occurs at the gap and rectangular domain. There is also a recirculation flow along the seat wall. The pressure values for the test model are also close to the reference values at the gap. In both cases the maximum pressure is about 190 bars, which occurs at the entrance of the gap (Fig. 3).

Case 1: 0.4 mm gap: Figure 4a and b show the velocity contours for conical and ball-head valves. As it can be seen in the graph in Fig. 4a, the velocity curve in the conical-head case has a sharper

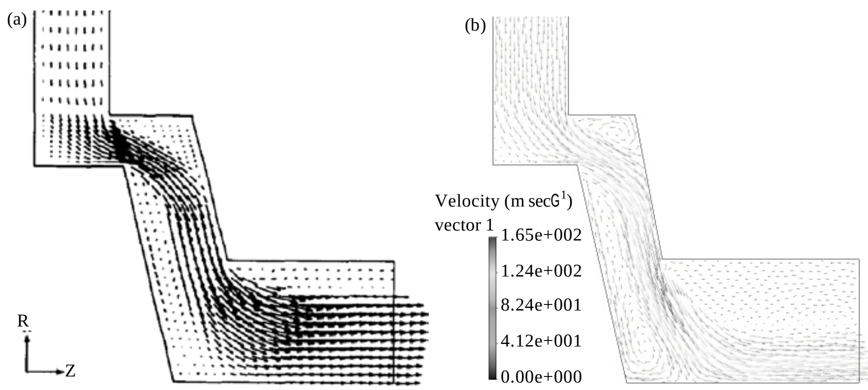

Fig. 3(a-b): Velocity vectors for the two cases, (a) Reference case and (b) The model

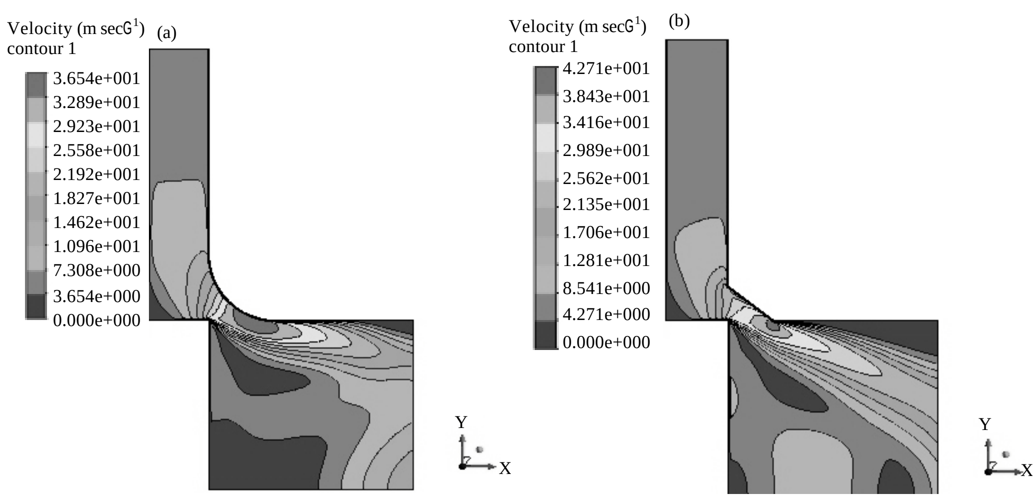

Fig. 4(a-b): Velocity contours for, (a) Ball-head and (b) conical head valve apex, which indicates that the velocity changes rapidly. The velocity value in the ball-head case tends to decrease more slowly and follows a gentler curve. The dominant parameter, which causes these differences in the velocity distribution, is the geometry of the valve head. In the spherical valve the velocity field changes direction along the valve head, while in the conical valve, the sharp corners of the valve head distract the velocity field and thus; the fluid velocity decreases. As is shown in Fig 4a, the high velocity region near the circular surface of the ball valve is larger, while in the conical valve this region is smaller but, has higher values, which are between 34-42 m sec^{-1}. The velocity along the conical valve changes rapidly, due to the shape of the valve head (Taghinia, 2011).

In the conical case the maximum velocity occurs at the second edge which is caused by the rapid change of pressure in that point. This rapid change of pressure also creates a reverse flow around the far edge of the conical valve. The pressure field in both cases experiences a significant flow recirculation near the vertical wall of the seat in rectangular domain which shows good agreement with previous studies in the literature (Passandideh-Fard and Moin, 2008).

There is also a relatively high vorticity near the ball head, which extends along the valve head. In the conical valve case the vorticity is present and it extends away from the valve surface.

Figure 5 shows the-Comparison of velocity values along valve head in two cases. These vortices play a key role in causing damage to the valve surface. Such reverse flows erode it and gradually make the valve head ineffective. Thus, the main attention is drawn to minimizing these negative factors when designing valves. The ball-head valve experiences a significant low pressure field in the circular part, which is caused by a high velocity field in that area. When the flow gets closer to the outlet, the pressure increases to adjust itself in the outlet condition.

The pressure varies through the "throat" area in a similar pattern in both cases, but in the rectangular domain the pressure jumps from -7.3×10^5 to -4.2×10^5 Pa, causing a pressure gradient in that area. This pressure gradient is the main reason for the recirculation flow in the rectangular domain (Fig. 6).

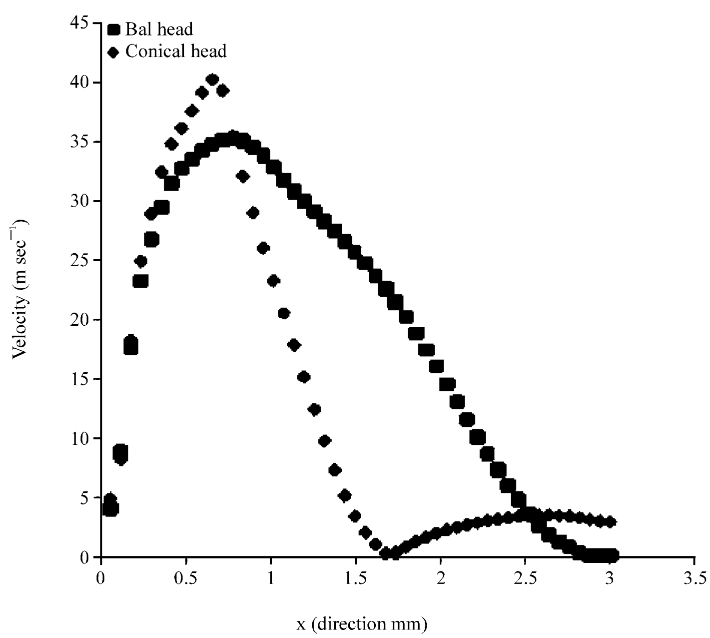

Fig. 5: Comparison of velocity values along valve head in two cases

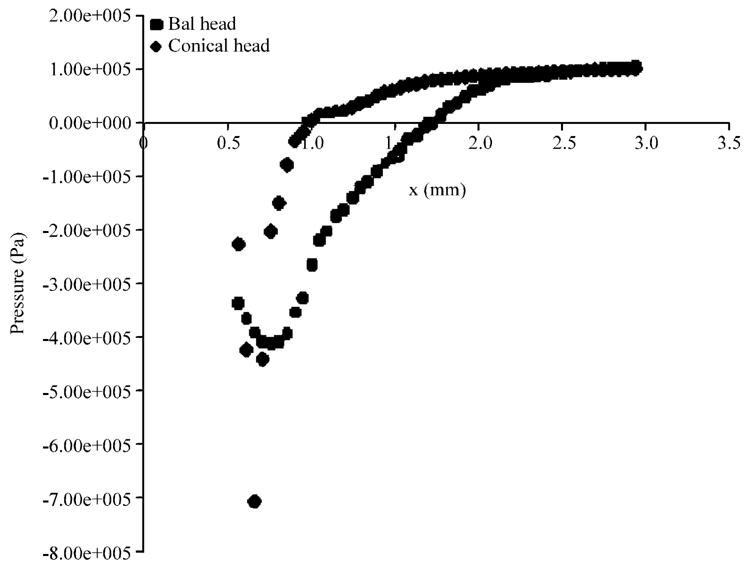

Fig. 6: Comparison of pressure values along the valve head

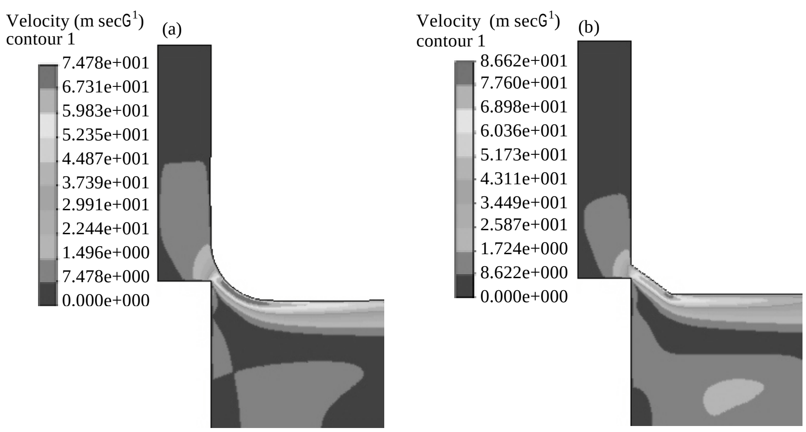

Fig. 7(a-b): Velocity of, (a) Ball-head and (b) Conical-head valve with 0.2 mm gap

Case 2: 0.2 mm gap: When the gap between the seat and the valve head is 0.2 mm, the 'throat' experiences a higher velocity field as compared to case 1. For the ball-head valve, similar to case 1, the high velocity field extends along the valve head. In both cases the maximum velocity occurs almost at the same location as in case 1. The maximum velocity magnitude is 86 m sec^{-1}, while in the ball-head case it is 74.7 m sec^{-1} (Fig. 7a). By comparing the velocity vectors, it can be noticed that there is a recirculation flow region at the center of the rectangular domain, which in the ball-head case is stronger and the velocity magnitude is lower in this area. This difference is due to the change of pressure at that location (Fig. 7b). Such recirculations have been seen in the works of Chen and Stoffel (2003) and Passandideh-Fard and Moin (2008) for the ball and conical head valves. Vorticity magnitudes for both valves are close to each other. A high vorticity can be seen around the seat and along the valve surface, which is caused by a sudden pressure drop at those locations. As the flow moves further from the seat, the vorticity magnitude starts to decrease until it reaches a stable value around the outlet.

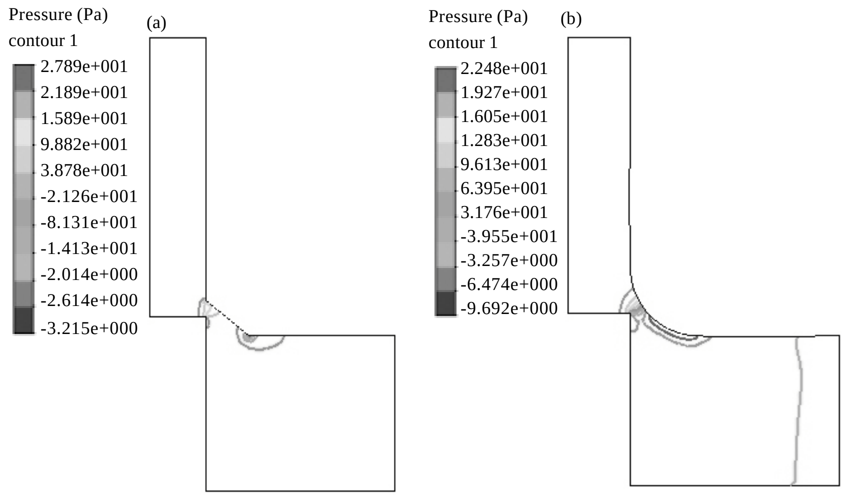

Fig. 8(a-b): Contours of pressure for valves head, head (a) Conical-head and (b) Ball-head with the 0.2 mm gap

The pressure distribution at the 'throat' is similar to case 1 for both shapes. For the ball-head valve, there is a relatively sudden change of pressure in comparison with case 1 (Fig. 8a). For the conical-head valve, the rectangular domain does not experience a significant change of pressure and tends to adjust to the pressure value at the outlet (Fig. 8b).

CONCLUSION

In this study, the flow characteristics have been investigated for ball and conical head valves with two different gap sizes. For this purpose a simple 2D axisymmetric geometry with the RNG k-ε turbulence model was applied. The results predicted a high velocity field at the gap of the valve, while in the ball-head valve case, the velocity field is higher. There is also a high reversed flow at the rectangular domain near the valve seat and the far side of the surface of the valve. The shape of the valve head has a dominant effect on the flow characteristics. A careful consideration of these parameters and flow patterns can be useful in designing water valves. For future studies, the model can be extended to a 3D one, thus allowing other flow characteristics such as cavitation to be investigated more precisely using a multiphase model.

REFERENCES

Chen, Q. and B. Stoffel, 2003. CFD simulation of a hydraulic conical valve by using cavitation model and simultaneously dynamic mesh modeling. Technische Universitat Darmstadt, Darmstadt, Germany.

Leino, T., K.T. Koskinen and M. Vilennius, 2003. CFD modeling of a water hydraulic poppet valve. Proceedings of the 8th Scandinavian International Conference on Fluid Power, Volume 1, May 7-9, 2003, Tampere, Finland, pp: 277-286.

Mokhtarzadeh-Dehghan, M.R., N. Ladommatos and T.J. Brennan, 1997. Finite element analysis of flow in a hydraulic pressure valve. J. Applied Math. Modell., 21: 437-445.

Passandideh-Fard, M. and H. Moin, 2008. A computational study of cavitation in a hydraulic poppet valve. Proceedings of the 5th International Conference on Transport Phenomena in Multiphase Systems, June 30-July 3, 2008, Bialystok, Poland, pp: 1-8.

Pounty, D.C., W. Weston and M.R. Banieghbal, 1989. A numerical study of turbulent flow characteristics of servo-valve orifices. Proc. Inst. Mech. Eng., 203: 139-147.

Sorensen, H.L., 1999. Experimental and numerical analysis of flow force compensation methods for hydraulic seat valve. Proceedings of the 6th Scandinavian International Conference on Fluid Power, May 26-28 1999, Tampere, Finland, pp: 471-482.

Taghinia, J., 2011. Fluid flow in hydraulic valves with ball and conical head. Report No. MECHA 13-2011, Department of Applied Mechanics, Aalto University, Sweden.

Tsukiji, T., Y. Yonezawa and Y. Ishii, 1995. Flow in a three dimensional poppet valve for oil hydraulic power applications. Trans. Jpn. Soc. Mech. Eng., 61: 998-1004.

Ueno, H., A. Okajima, H. Tanaka and T. Hasegawa, 1994. Noise measurement and numerical simulation of oil flow in pressure control valves. JSME Int. J. Ser. B, 37: 336-341.

Vaughan, N.D., D.N. Johnson and K.A. Edge, 1992. Numerical simulation of fluid flow in poppet valves. Proc. Inst. Mech. Eng. Part C: J. Mech. Eng. Sci., 206: 119-127.

Yakhot, V. and S.A. Orszag, 1986. Renormalization group analysis of turbulence: I-basic theory. J. Sci. Comput., 1: 3-51.