Materials Research Proceedings **5** (2018) 1-6 doi: http://dx.doi.org/10.21741/9781945291814-1

Simulation of Cavitation Erosion in a Hydraulic Valve

DOMAGAŁA Mariusz[1, a *], MOMENI Hassan[2,b], DOMAGALA-FABIS Joanna[1,c]
FILO Grzegorz[1,d], KWIATKOWSKI Dominik[1,e]

[1]Cracow University of Technology, Institute of Applied Informatics, Al. Jana Pawla II 37
31-841 Krakow, Poland

[2]Western Norway University of Applied Sciences, Department of Mechanical and Marine Engineering, Postboks 7030, 5020 Bergen, Norway

[a]domagala@mech.pk.edu.pl, [b]Hassan.Momeni@hvl.no, [c]fabis@mech.pk.edu.pl,
[d]filo@mech.pk.edu.pl, [e]dkwiatkowski@pk.edu.pl

Keywords: Cavitation Erosion, CFD Simulation, Non-return Valve

Abstract. This paper presents a numerical simulation of cavitation of a non-return hydraulic valve. Cavitation is one of the most common reasons of hydraulic system failures or malfunctioning. Due to complex geometry and flow path which may rapidly change, hydraulic valves are exposed to a cavitation phenomenon. Additionally, hydraulic fluid always includes contaminants including dissolver air, which may accelerate nucleation formation. Despite the fact that cavitation is not a fully discovered phenomenon, an attempt of simulation of cavitation process was taken using computational fluid dynamics (CFD) tools.

Introduction

Cavitation as a phenomenon has been studied from the early beginning of XX century, started by Rayleigh. Despite extensive research and numerous investigations on cavitation, it has not been completely discovered yet. The most common explanation of this phenomenon is that cavitation arises during fluid flow when pressure falls below critical values, which is called saturation pressure. It might be observed during internal flow when fluids gather locally velocity causing local pressure drop where bubbles of fluid vapor and air are created. These bubbles collapse when fluid reaches areas with higher pressure causing bubbles implosions and high peak pressure around bubbles surroundings. Implosions and high pressure peaks lead to noise, mechanical vibrations and surface erosion. Attempts of mathematical description of cavitation and nucleation formation have been conducted for many years [1]. Research on cavitation in hydraulic components was conducted by simulations verified by experimental tests or in an experimental way [2], which is difficult due to problems with direct observation of the phenomenon. R. Amirante, et al. [3] and J. R. Valdés at al. [4] conducted numerical simulations and experimental tests on hydraulic valves.

Erosion caused by cavitation in a hydraulic valve is very well known. Long term exposure of flow to cavitation makes valve surfaces erode, which leads to improper operation of the valve and, in the consequence, malfunctioning of the hydraulic system. An example of cavitation erosion is presented in Fig. 1.

Simulation of cavitation in hydraulic liquids is not an easy task. Contrary to water, which is a single compound liquid, hydraulic fluids are a mixture of mineral oils, esters, corrosion inhibitors and other additives and therefore it is difficult to evaluate critical saturation pressure. This paper presents a simulation of cavitation in a no-return hydraulic ball type valve. To this purpose, a general purpose CFD code, Ansys CFX, was used with an implemented two phase flow with phase change.

Materials Research Proceedings **5** (2018) 1-6 doi: http://dx.doi.org/10.21741/9781945291814-1

Fig. 1. Eroded surface of a valve [5]

Cavitation model

As it was mentioned before, lots of studies have been conducted on cavitation and bubble formation. Generally, the tendency of fluid flow to cavitation may be expressed as [6]:

$$c_a = \frac{p - p_v}{0.5 \rho U^2} \quad (1)$$

The Rayleigh-Plesset formula is one of the approaches which describe bubble dynamics:

$$R_b \frac{d^2 R_b}{dt^2} + \frac{3}{2}\left(\frac{dR_b}{dt}\right)^2 + \frac{2\sigma}{\rho R_b} = \frac{p_v - p}{\rho} \quad (2)$$

After neglecting surface tension and term of second order, the above eq. has the following form:

$$\frac{dR_b}{dt} = \sqrt{\frac{2}{3}\left(\frac{p_v - p}{\rho}\right)} \quad (3)$$

The rate of changes of bubble volume is expressed by the following formula:

$$\frac{dV_b}{dt} = 4\pi {R_b}^2 \sqrt{\frac{2}{3}\left(\frac{p_v - p}{\rho}\right)} \quad (4)$$

The rate of changes of bubble mass is expressed by:

$$\frac{dm_g}{dt} = 4\pi {R_b}^2 \rho_g \sqrt{\frac{2}{3}\left(\frac{p_v - p}{\rho}\right)} \quad (5)$$

The number of bubbles *Nb* per unit volume rg is expressed by:

$$r_g = \frac{4}{3}\pi R_b^2 N_b \quad (6)$$

The total interphase mass transfer per unit volume is expressed by:

$$\dot{m}_{fg} = 3\frac{r_g \rho_g}{R_b}\sqrt{\frac{2}{3}\frac{p_v - p}{p_f}} \quad (7)$$

After including condensation to the above expression, it has the following form:

Materials Research Proceedings **5** (2018) 1-6 doi: http://dx.doi.org/10.21741/9781945291814-1

$$\dot{m}_{fg} = 3F\frac{r_g\rho_g}{R_b}\sqrt{\frac{2}{3}\frac{|p_v-p|}{p_f}}\,\mathrm{sgn}(p_v-p) \tag{8}$$

And finally, the vapor transport equation appears in the following form:

$$\frac{\partial}{\partial t}(\alpha\rho_v) + \nabla\cdot\left(\alpha\rho_v\mathbf{U}_f\right) = R_e - R_c \tag{9}$$

where:

R_b – bubble radius, p_v– vapor pressure, p – pressure of liquid surrounding the bubble, ρ–liquid density, ρ_g– vapor density, σ – surface tension, $\mathbf{U}_v$ – vapor phase velocity, α– vapor volume fraction, n– bubble number, R– phase change rate, f_v– vapor mass fraction, f_g– non-condensable gases, R_e, R_c- mass transfer source terms connected to the growth and collapse of the vapor bubbles, respectively.

Object of study

A non-return valve which is also called a check or a one way valve is the object of the study. The aim of such a valve is to provide flow only in one direction and secure against reverse flow.

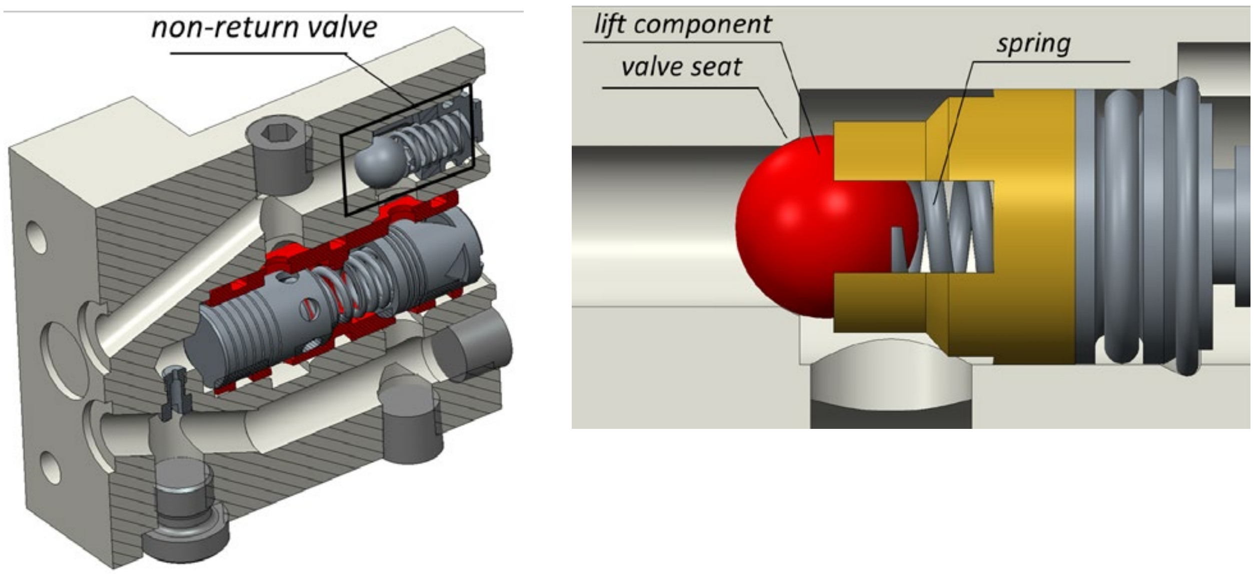

Fig. 2. Non-return valve assembled in a flow control valve

It consists a lift component, which has a spherical or conical shape and a spring which sets the opening pressure value. Such valves might be installed directly in-line or as a component of other valves.

CFD simulations

A flow simulation was conducted in the Ansys CFX general purpose CFD code, which uses the Finite Volume Method (FVM) for solving flow governing equations and applies a multiphase flow model with a phase change. For the numerical simulation, half of a valve model was used. Boundary conditions and a grid is presented in Fig. 3.

Materials Research Proceedings **5** (2018) 1-6 doi: http://dx.doi.org/10.21741/9781945291814-1

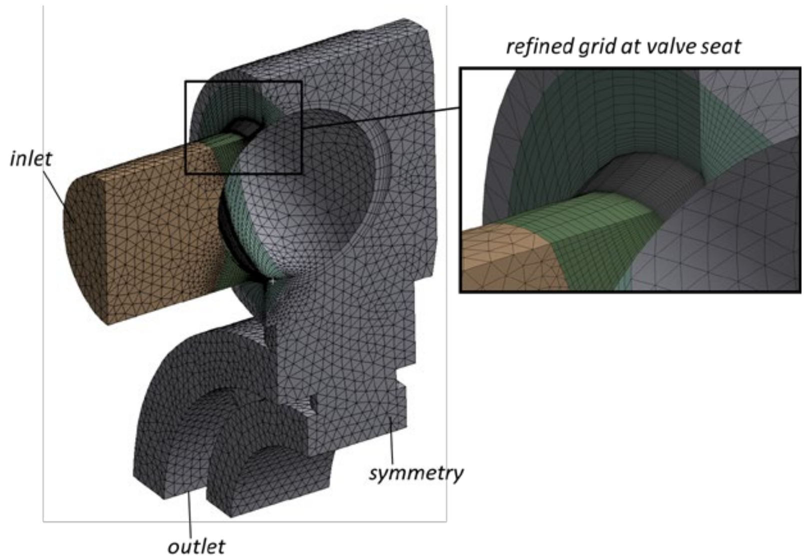

Fig. 3. CFD model: grid and boundary conditions

The CFD simulation was performed for a fixed ball position for steady state conditions and for the following assumptions:

- Fluid (hydraulic oil) has the following constant properties: density 880 [kg/m^3], viscosity υ=40 [mm^2/s]
- Flow is turbulent: k-ϖ turbulence model was used,
- Model is in thermodynamic equilibrium, heat transfer is not included.
- Saturation pressure is set to 0.2 [bar] [7],
- Simulation includes two phases flow with a phase change.
- Fluid is homogeneous without any dissolved gases.
- Half of a geometrical model was used in simulations.
- Simulations were performed for two cases, which differs the angle of the valve seat.

The CFD results, as a fluid velocity distribution for both cases at a symmetry plane are presented in Fig. 4.

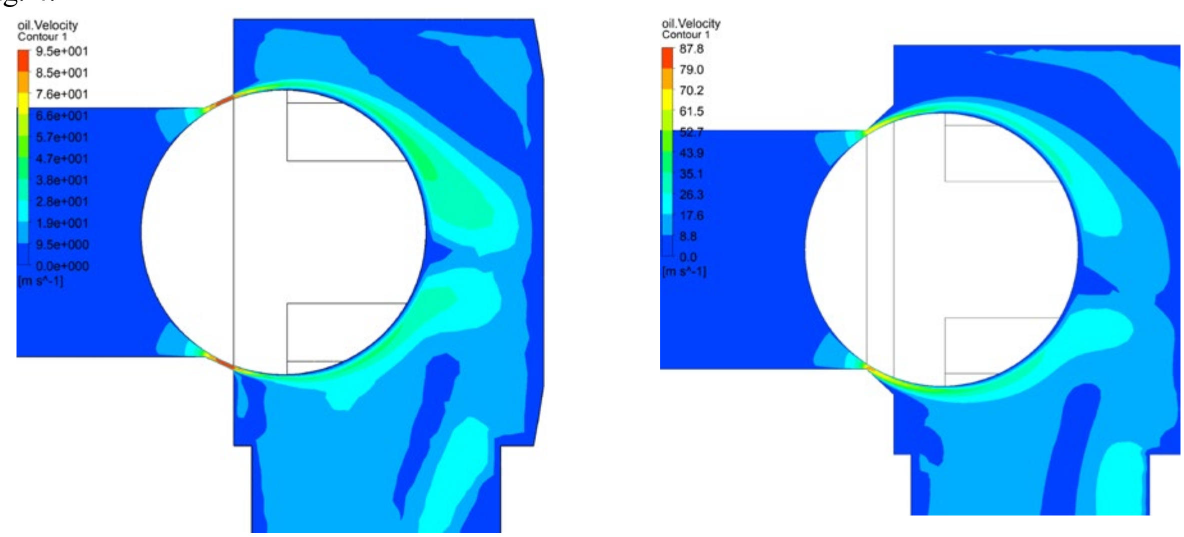

Fig. 4. Fluid velocity at a symmetry plane Velocity distribution shows that the angle of a valve seat influences the maximal velocity of the fluid.

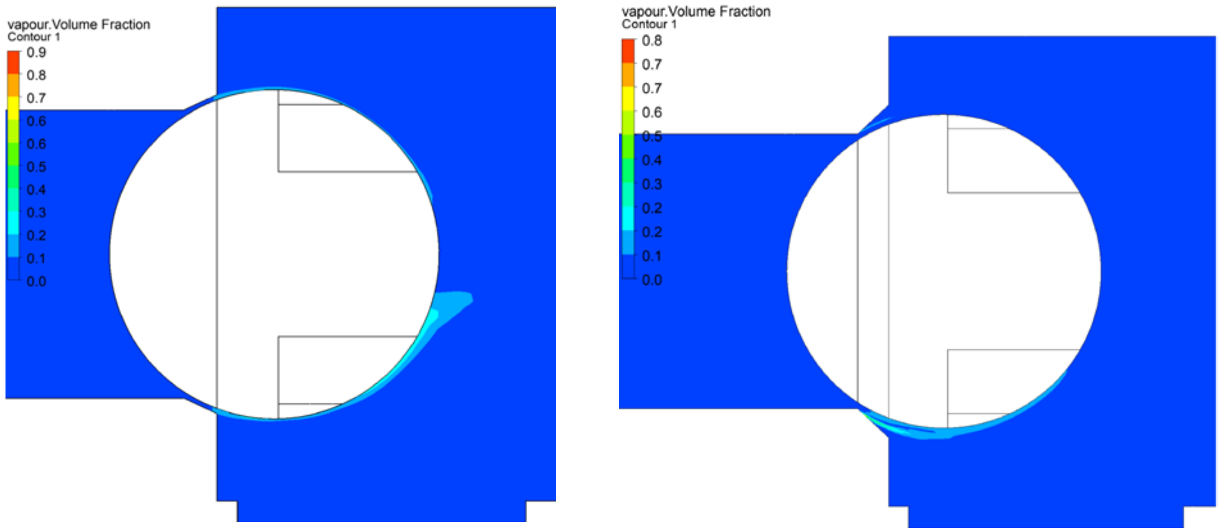

Fig. 5. Gaseous volume fraction

Figure 5 shows a gaseous volume fraction which allows the identification of areas where cavitation may occur. As it was expected, the area with a sudden velocity change is the most exposed to cavitation occurrence. However, the design parameters of the valve (angle of the seat) have influence on the place where cavitation may occur. Figure 6 shows path lines and areas where gaseous fraction may appear during fluid flow.

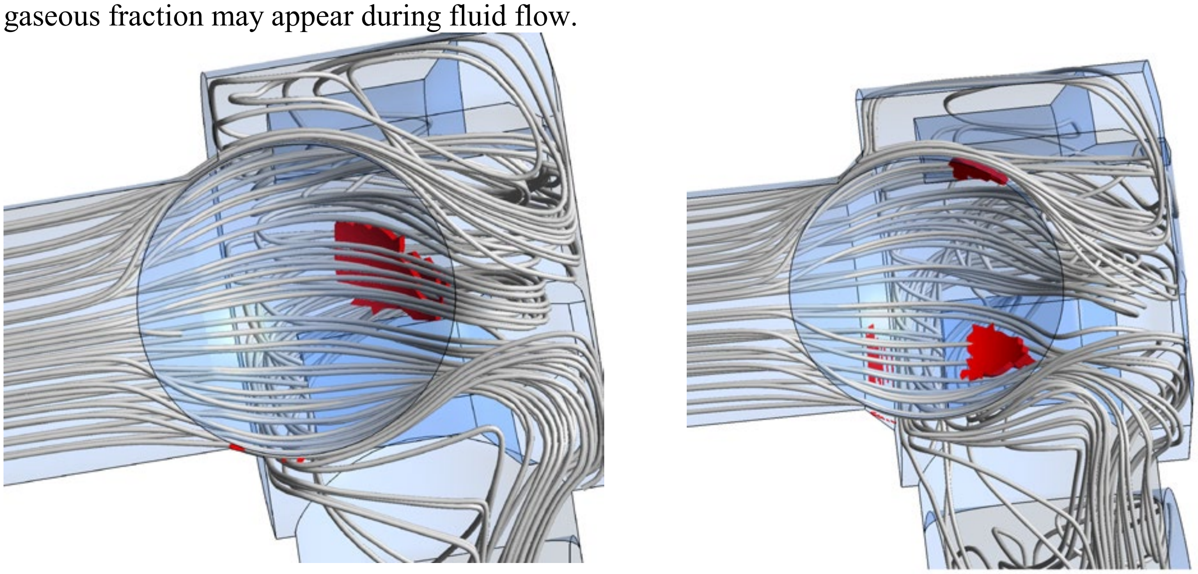

Fig. 6. Path lines and gaseous volume fraction

Summary

This paper presents a simulation of cavitation for one of the most commonly used hydraulic component, which is a ball type non-return valve. Simulations were performed for two cases, differing with valve seat geometry in the Ansys CFX code as a two phase flow with a phase change. The obtained results allowed us to indicate where cavitation may appear and what influence a valve design may have on cavitation phenomenon. The results presented in this paper showed that CFD codes may be recognized as tools for predicting cavitation erosion.

References

[1] Brennen C, E, Cavitation and buble dynamics. Oxford University Press (1995).

[2] Z. Kudzma, M. Stasiak, Studies of flow and cavitation of hydraulic lift valve, Archives of civil and mechanical engineering 15, 2015, 951-961. https://doi.org/10.1016/j.acme.2015.05.003

[3] R. Amirante, E. Distaso, P. Tamburrano, Experimental and numerical analysis of cavitation in hydraulic proportional directional valves, Energy Conversion and Management, 87 (2014) 208–219. https://doi.org/10.1016/j.enconman.2014.07.031

[4] J. R. Valdés, J. M. Rodríguez, R. Monge a, J. C. Peña, T. Pütz, Numerical simulation and experimental validation of the cavitating flow through a ball check valve, Energy Conversion and Management 78 (2014) 776–786. https://doi.org/10.1016/j.enconman.2013.11.038

[5] http://valvemagazine.com: Flashing and cavitation (information available on 15.02.2018)

[6] ANSYS CFX-Solver Theory Guide, Release 18.2, Ansys Inc, 2017

[7] M. Schümichen, F. Rüdiger, J. Fröhlich, Simulation of the cavitating flow in a model oil hydraulic spool valve using different model approaches 10th International Fluid Power Conference (10. IFK) March 8 - 10, 2016, Vol. 1, pp. 321-332

[8] J. Fabis-Domagala, Analysis of dfefects and failures of hydraulic gear pumps with the use of selected qualitative methods, Tribologia 2/2017, 33-38, ISSN 0208-7774

[9] M. Domagala, Simulation of cavitation in jet pumps, Technical Transactions, ISSN 1897-6328, 2013, 51-58

[10] E. Lisowski, G. Filo, CFD analysis of proportional flow control valve with an innovative opening shape, Energy Conversion and Management 123, 2016, 15-28. https://doi.org/10.1016/j.enconman.2016.06.025