

Determination of Flow Parameters through CFD Analysis for Agricultural Irrigation Equipment: A Case Study for a Mini Valve

H. Kursat CELIK¹, Allan E. W. RENNIE², Davut KARAYEL¹, Ibrahim AKINCI¹

¹ Department of Agricultural Machinery, Faculty of Agriculture, Akdeniz University, Antalya, TURKEY

² Lancaster Product Development Unit, Engineering Department, Lancaster University, Lancaster, UK
hkcelik@akdeniz.edu.tr

Received (Geliş Tarihi): 08.05.2011

Accepted (Kabul Tarihi): 09.07.2011

Abstract: Computational Fluid Dynamics (CFD) is a powerful technique, which has been used successfully for a number of years in a wide variety of engineering disciplines, to develop an approach for solving complex fluid flow problems. In more recent years, with the desire to provide more efficient operations, it has become increasingly important to adapt this analysis methodology for advanced agricultural engineering design applications. This paper presents a case study for determining a number of flow parameters for a mini valve, which is used in agricultural irrigation systems, through the utilisation of CFD techniques. In the study, a sample plastic mini valve has been utilised for CFD analyses. Flow behaviour of the fluid in the valve was simulated three-dimensionally using a commercial CFD code. Pressure loss, head loss, flow coefficient, resistance coefficient and cavitation index parameters were calculated for different flow rates with different valve opening positions with the aid of simulation results. According to CFD results; maximum pressure loss of 8214.47 Pa was obtained in the analysis which was set up with flow rate of 4 L min⁻¹ at 45° valve opening positions. And the pressure loss is increasing if valve opening angle and inlet flow volume rates are increased. The Head loss and resistance coefficient are also increasing but flow coefficient is decreasing. The valve opening positions are affecting to the pressure and velocity distributions. In addition to this, charts have been provided to evaluate variations of the pressure, velocity and the flow parameters for the valve. This research improved the understanding of the fluid flow performance of the irrigation equipment and contributes to further research into the development of agricultural irrigation equipment enabled through the utilisation of advanced computer aided engineering (CAE) tools.

Keywords: Agricultural Engineering Design, CAE, CFD, Flow Parameters, Irrigation Valve

INTRODUCTION

Computational fluid dynamics (CFD) is a powerful tool, which is utilised to solve fluid flow application problems based on numerical methods in very wide range of engineering disciplines. Before the introduction of advanced computer aided engineering (CAE) applications, it was not easy to investigate details of the fluid flow inside or outside of an object. It is now possible to examine and predict flow characteristics in existing product geometries or to optimise the geometry for best flow paths. The numerical method based fluid flow simulations provide essential indications during the preliminary phase and allows compression of the time of development and the costs of the subsequent experimental analysis and pre-manufacturing

processes (Min et al., 2008). Hence, the approach of using CFD techniques has been substantially appreciated in mainstream scientific research and industrial engineering communities (Song and Park, 2007). This wide range of CFD simulation usage in several engineering fluid flow applications shows that it is also very important to use these techniques in agricultural irrigation equipment design processes to gain an understanding and appreciation of the advantages of advanced CAE applications.

Plastic mini valves are widely used in irrigation equipment in agriculture, with various types of these valves used in the field. The main aim of these types of valves is to control the volumetric flow rate of water for the related lateral pipes in an irrigation

system. Even though these plastic mini valves are not operated under high value pressures, their geometrical shapes, part thicknesses, constituent materials and surface roughness as a result of the manufacturing methods used, have important roles on the flow parameters which are partly affected by the total irrigation system performance. Hence, it is very important to understand the flow behaviours and determine flow parameters such as pressure loss, head loss, flow coefficient, resistance coefficient and cavitation index which are the key requirements for the design of the valves. Most especially, the cavitation, which may cause noise and vibrations, due to local low pressure, has to be understood well during operation of the valves (Chern et al., 2007). In any flow system, the familiar restrictors are the valve and hence its design and performance analysis will be a significant task (Prakash et al., 2009).

There are many reports on the determination of flow parameters and the design of various valves using CFD simulations and experimental techniques (Mokhtarzadeh Dehghan et al., 1997; Forsberg and Persson, 2005; Chern et al., 2007; Song and Park, 2007; Min et al., 2008; Prakash et al., 2009; Song et al, 2009; Lee et al., 2010a; Lee et al., 2010b; Kabwe et al., 2010), however, studies on the design and determination of the flow parameters of agricultural irrigation equipment using CFD simulations are limited.

The main objective of the case study in this paper is to determine related pressure loss and flow parameters using CFD simulation techniques which can prove advantageous for the agricultural irrigation equipment design and manufacturing industry. In this regard, a sample plastic mini valve is utilised for CFD analyses. Flow behaviour of the fluid in the valve was simulated three-dimensionally (3-D) using a commercial CFD code. In accordance with the simulation results, some of the flow parameters such as pressure loss, head loss, flow coefficient, resistance coefficient and cavitation index were calculated for different flow volume rates with different valve opening positions. In addition to this, charts have been provided to evaluate variations of the pressure, velocity and flow parameters for working conditions of the valve.

MATERIALS and METHODS

Flow Parameters

In engineering practice, the loss during fluid flow in any piping system is traditionally split into two components: the loss due to friction along straight pipe sections and the loss due to localised pipe features such as bends, various cocks, valves, throttles etc.(SolidWorks Product, 2010a). Generally, total loss and related coefficients can be calculated by formulae based on theoretical and experimental investigations for the whole system but the other matter is determining the local losses and local values of the related coefficients which are difficult to predict and calculate experimentally. At these points, flow simulations can be an alternative approach to the traditional problems associated with predicting this kind of local loss computationally with good accuracy for any desired point of the system.

In general, a valve is evaluated using three coefficients depending on fluid properties, flow velocity and pressure differences (Pressure loss: ΔP) between inlet and outlet pressures: flow coefficient, resistance (loss) coefficient and cavitation index (Chern et al, 2007). In addition to this, flow simulation applications let researchers calculate other parameters such as pressure loss, head loss etc. automatically which can aid the evaluation process.

Valves may be regarded as analogous to control orifices in which the area of opening is readily adjustable. As such, the friction loss across the valve varies with flow, as expressed by the general relationship in Equations 1 and 2 (Smith and Zappe, 2004).

$$V \propto (\Delta H)^{1/2} \quad (1)$$

$$V \propto (\Delta P)^{1/2} \quad (2)$$

where, V is flow velocity, ΔH is head loss, ΔP is pressure loss.

For any valve position, numerous relationships between flow and flow resistance have been established, using determined resistance or flow parameters. The parameters which were used to relate calculations in this paper based on CFD analysis results are explained below.

Flow coefficient (A_v)

The flow coefficient (A_v) states the number of cubic metres per second of water at a temperature between 5° and 40° C that will flow through the valve with a pressure loss of one Pascal at a specific opening position, as defined by Equation 3 below (in coherent SI units) (Smith and Zappe, 2004).

$$A_v = Q \left(\frac{\rho}{\Delta P} \right)^{1/2} \quad (3)$$

where, Q is flow volume rate (m^3/s), ρ is density of the water (kg/m^3), ΔP is pressure loss [Pa].

Resistance (loss / pressure) coefficient (δ)

The resistance coefficient (δ) defines the friction loss attributable to a valve in a pipeline in terms of velocity-pressure as expressed by Equation 4 (in coherent SI units) (Smith and Zappe, 2004; Chern et al., 2007; Kabwe et al., 2010;).

$$\delta = \frac{2\Delta P}{\rho V^2} \quad (4)$$

where, ΔP is pressure loss (Pa), ρ is density of the water (kg/m^3), V is flow velocity at inlet (m/s).

Cavitation index (σ)

Cavitation is the sudden formation and collapse (vaporisation) of a liquid downstream of the valve due to localised low pressure zones. The cavitation index (σ) is used to predict the conditions where cavitation happens. This parameter is presented in literature in various forms. The following expression was used in this paper as a convenient equation to investigate cavitation condition by comparing resistance coefficient and is denoted by Equation 5 (Falvey, 1990; Ivanell, 2001; Smith and Zappe, 2004; Chern et al, 2007; Nicholls and Turnock, 2007; Schiavello and Visser, 2008; Baldygaa et al., 2009; Val-matic, 2010).

$$\sigma = \frac{P_{Total} - P_v}{0.5 \rho V^2} \quad (5)$$

where, P_{Total} is total pressure at measured point (absolute) [Pa], P_v is saturated vapour pressure [Pa] (2339 Pa at 20° C), ρ is density of the water

(kg/m^3), V is fluid velocity at the measured point (m/s).

Where there is a large cavitation index, this means that there is a small probability that cavitation will occur. The model will not be able to follow air bubbles, as it is a one-phase flow, but the source of cavitation can be detected by checking whether the measured pressure reaches the evaporating point, according to the following conditions (Falvey, 1990; Ivanell, 2001).

$$\sigma \geq |\delta| \Leftrightarrow P \geq P_v \Rightarrow \text{No cavitation} \quad (6)$$

$$\sigma < |\delta| \Leftrightarrow P < P_v \Rightarrow \text{Cavitation appears} \quad (7)$$

Head loss (ΔH)

Head loss is a measure of the reduction in the total head (sum of elevation head, velocity head and pressure head) of the fluid as it moves through a fluid system. In the literature, the most used method is Darcy-Weisbach equations to calculate head loss in a horizontal pipe and derived equation from Darcy-Weisbach equations for head loss can be expressed as per Equation 8 (Recktenwald, 2007, Lee et al, 2010a).

$$\Delta H = \frac{\Delta P}{\rho g} \quad [\text{m}] \quad (8)$$

where, ΔP is pressure loss [Pa], ΔH is Head Loss [m], ρ is density of the water [kg m^{-3}], earth gravity [9.81 m s^{-2}].

CFD Geometry

The ultimate goal of the utilisation of CFD is to understand the physical events that occur in the flow of fluids around and within designated objects (Lomax et al, 1999). For the CFD analysis, the plastic mini valve, which had nominal dimensions of 16 x 16 mm (inlet and outlet pipe diameters) was reverse engineered using SolidWorks 3-D parametric solid modelling design software in accordance with its original dimensions. Upstream and downstream pipes were assembled to the valve solid model by a factor of two times the pipe nominal diameter (D) before valve and eight times the nominal pipe diameter

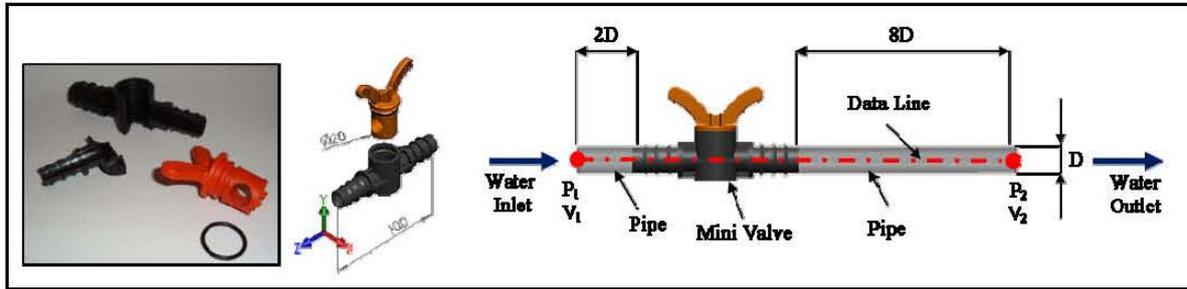


Figure 1. Plastic mini valve, its 3-D CFD Geometry Model and data line

after (Song and Park, 2007; Lee et al., 2010a). In addition to this, a data line with total length of 258 mm is prepared to measure the flow data at any point of the flow path inside the pipe and valve assembly. The plastic mini valve, its CFD geometry model and data line are shown in Figure 1.

CFD Analysis

In the case study, SolidWorks Flow Simulation commercial CFD code was utilised to simulate 3-D flow behaviour inside the valve assembly. To simulate the flow situation, it is necessary to solve Navier-Stokes equations for a 3-D model. The CFD code solves Navier-Stokes equations, which are formulations of mass, momentum, and energy conservation laws for fluid flows (Equations 9, 10, 11) (SolidWorks Product, 2010b).

The momentum conservation equation:

$$\frac{\partial}{\partial t}(\rho v_i) + \frac{\partial}{\partial x_j}(\rho v_i v_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i \quad (9)$$

The mass conservation equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho v_i) = S_m \quad (10)$$

The energy conservation equation:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}(v_i(\rho E + P)) = \frac{\partial}{\partial x_i}[k_{eff} \frac{\partial T}{\partial x_i} - \sum_j h_j J_j + v_j(\tau_{ij})_{eff}] + S_h \quad (11)$$

where, P is static pressure, v is velocity component, τ_{ij} is stress tensor, ρ is density of fluid, and g_i and F_i are the gravitational body force and the external body forces.

In fact, even the simulations give very good prediction points for real life responses; it is true to say that it is nearly impossible to catch all real life responses one-by-one from simulations due to the technological limitations and unpredictability of the material, fluid, dynamic environment conditions, etc., however, some assumptions have to be made within the simulation set-up procedures to gain approximate solutions. Hence, some assumptions have been made in this case study.

CFD analyses were carried out for an inlet flow volume rate of 2, 3 and 4 L/min with 0°, 15°, 30° and 45° valve openings respectively for each. Related flow parameters were calculated for these different flow rates with different valve opening positions with the aid of simulation results data obtained from the data line and surfaces of water inlet and water outlet. Identical boundary conditions were defined for each analysis of the valve. In the analyses, the valve parts are assumed rigid (there will not be any physical deflection as a result of pressure during fluid flow and there will be no leakage at any point of the assembly connections). Water was assigned as the fluid. Viscous and incompressible flow was assumed at environment temperature of 20° C. Earth gravity of 9.81 m/s² was considered for the simulation and surface roughness was assigned as 0.01 mm for the walls of the solid models. Default mesh function with advanced channel refinement option and minimum fluid gap size of 4 mm were applied in the software for the cell structure of the fluid domain. For most flows, it is difficult to have a good estimation of the turbulence so it is recommended that the default automatic turbulence parameters be used in SolidWorks Flow Simulation software. Hence, the laminar and turbulent default flow condition

parameters were accepted in the software for the CFD analysis of the valve (SolidWorks Product, 2010b). The whole calculation domain includes the valve and the pipe with a rectangular volume of 258.5 x 16.75 x 21.42 mm (maximum fluid domain for the case of a valve opening of 45°). Environment pressure (absolute pressure) of 101,325 Pa was defined for the valve's outlet holes. Fluid volumes which are considered in the CFD analysis with defined valve openings are shown in Figure 2.

RESULTS and DISCUSSION

A total of twelve analyses were carried out. All results have been obtained visually and numerically after solving through the CFD analyses. CFD simulations provided the ability to undertake advanced investigations visually for fluid flow

behaviour inside the valve at any chosen point. As a sample analysis, in order to understand the implications at the highest pressure effect, the case was set up with an inlet flow volume rate of 4 L min⁻¹ at 45° valve opening position, with the resultant plots for total pressure and velocity distributions shown in Figure 3.

In this analysis, maximum total pressure of 112557 Pa (absolute) and maximum velocity of 4.37 m s⁻¹ were obtained for the whole model. For the flow parameters calculation, data taken from the data line were used. According to these calculations, a pressure loss of 8214.47 Pa, head loss of 0.84 m, flow coefficient of 0.00001, resistance coefficient of 86.74 and cavitation index of 791.06 were calculated. The charts in Figure 4 are given to investigate changing of the parameters locally point by point at the data line.

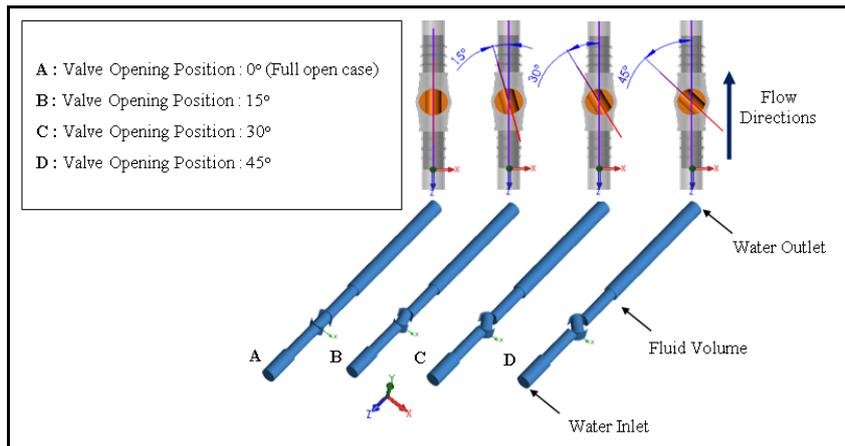


Figure 2. Valve opening positions and CFD fluid calculation volumes

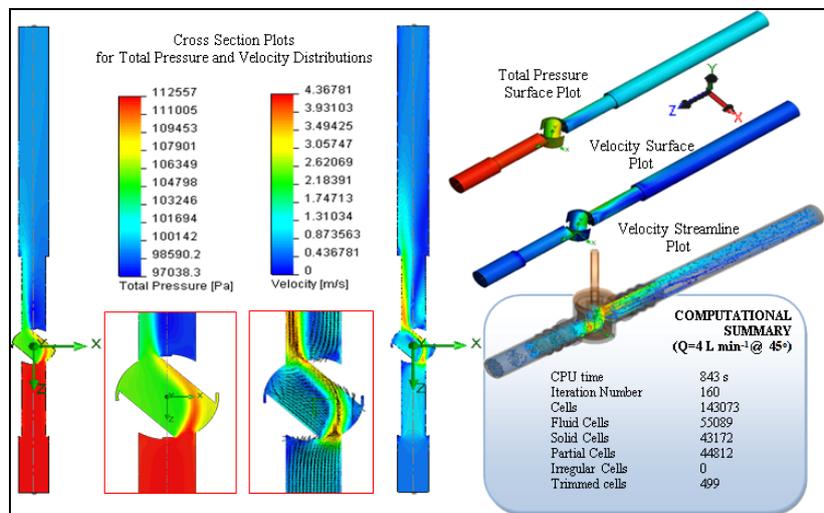


Figure 3. CFD Analysis result plots (Q = 4 L/min at 45°)

In the investigation of the valve performance, it is important to make comparisons between analyses to see how defined conditions affect flow parameters such as flow volume rates and valve opening positions. In accordance with the results of all twelve

of the analyses undertaken, the calculated flow parameters are given in Table 1. Total pressure distribution plots and charted flow parameters are shown in Figure 5 and Figure 6 respectively.

Table 1. Comparison Table for Calculated Flow Parameters from Data Line

Flow Volume at Valve Opening Angle (Degree)	ΔP [Pa] (Pressure Loss)	ΔH [m] (Head Loss)	A_v [-] (Flow Coefficient)	δ [-] (Resistance Coefficient)	σ [-] (Cavitation Index)
Q = 2 L min ⁻¹ @ 0 °	55.29	0.00565	0.00014	2.33400	2662.30832
Q = 2 L min ⁻¹ @ 15 °	93.42	0.00955	0.00011	3.94329	2759.52378
Q = 2 L min ⁻¹ @ 30 °	414.98	0.04241	0.00005	17.50294	2767.29588
Q = 2 L min ⁻¹ @ 45 °	2062.32	0.21074	0.00002	86.92796	2845.87626
Q = 3 L min ⁻¹ @ 0 °	112.59	0.01150	0.00010	2.11445	1250.73832
Q = 3 L min ⁻¹ @ 15 °	202.69	0.02071	0.00007	3.80627	1301.63707
Q = 3 L min ⁻¹ @ 30 °	928.18	0.09485	0.00003	17.41719	1331.25372
Q = 3 L min ⁻¹ @ 45 °	4634.22	0.47355	0.00002	86.90987	1364.61920
Q = 4 L min ⁻¹ @ 0 °	192.26	0.01965	0.00008	2.03278	729.42800
Q = 4 L min ⁻¹ @ 15 °	357.53	0.03653	0.00006	3.78008	756.86952
Q = 4 L min ⁻¹ @ 30 °	1645.79	0.16818	0.00003	17.38624	770.83287
Q = 4 L min ⁻¹ @ 45 °	8214.47	0.83940	0.00001	86.73826	791.06392

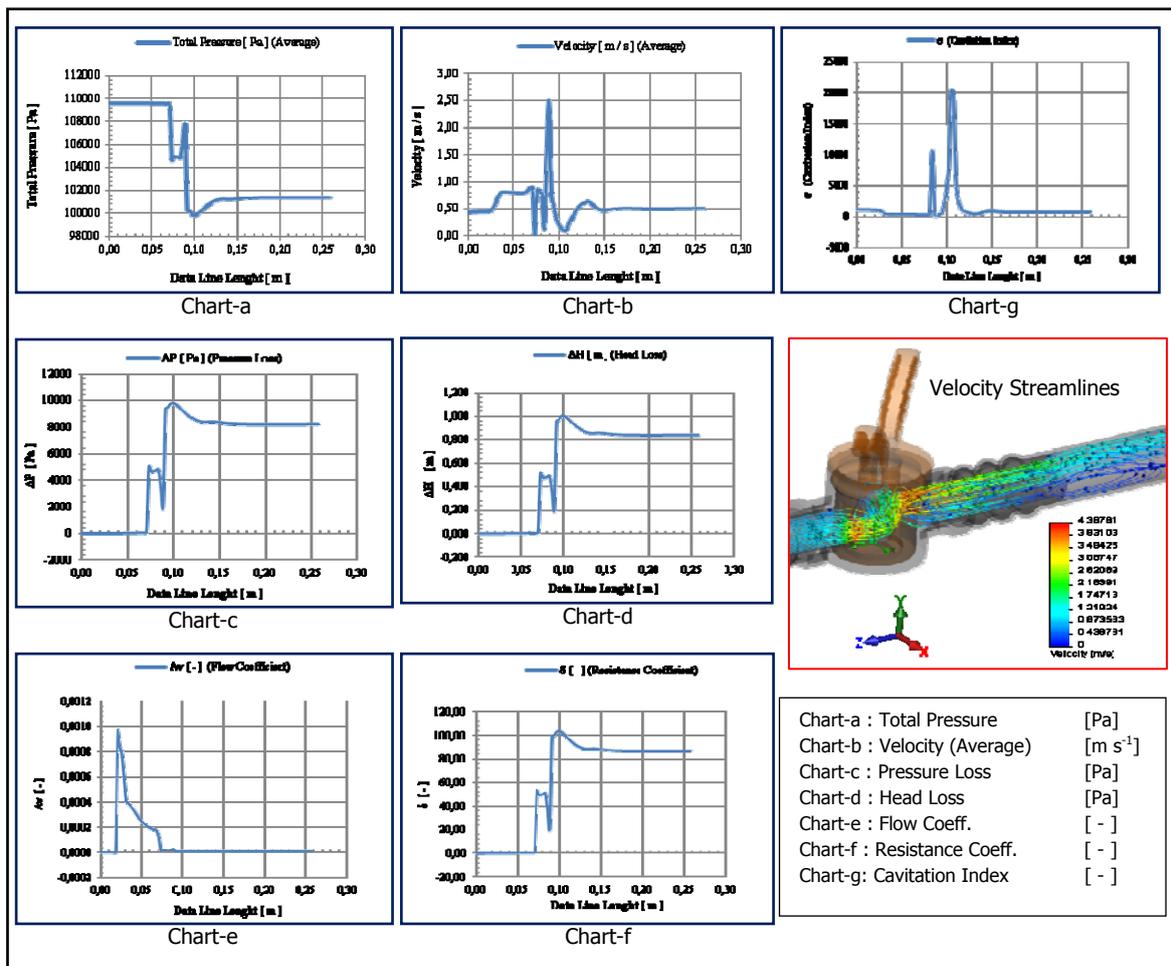


Figure 4. Charts for calculated flow parameters from data line (Q = 4 L/min at 45°)

At first sight in the evaluation, pressure loss is very obvious due to resistance of the valve and it is clear to see that the pressure loss is increasing if valve opening angle and inlet flow volume rates are increased. In parallel to this situation, head loss and resistance coefficient are increasing but flow coefficient is decreasing. In addition to this, it can be concluded that pressure and velocity distributions change differently from the upstream line to the

downstream line according to the valve opening positions. This is because the flow path changes. If the cavitation index is evaluated for each analysis, it can be seen that the values are quite high. However, according to the definitions in Equations 6 and 7, it can be said that the probability of cavitation is not likely, but the cavitation risk will increase with increasing flow volume rate.

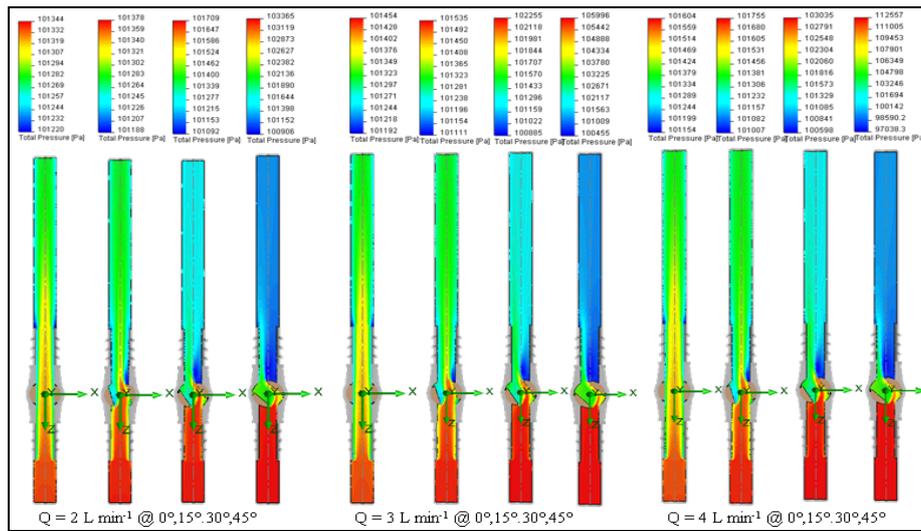


Figure 5. Total Pressure distribution for different flow volume rates and valve openings

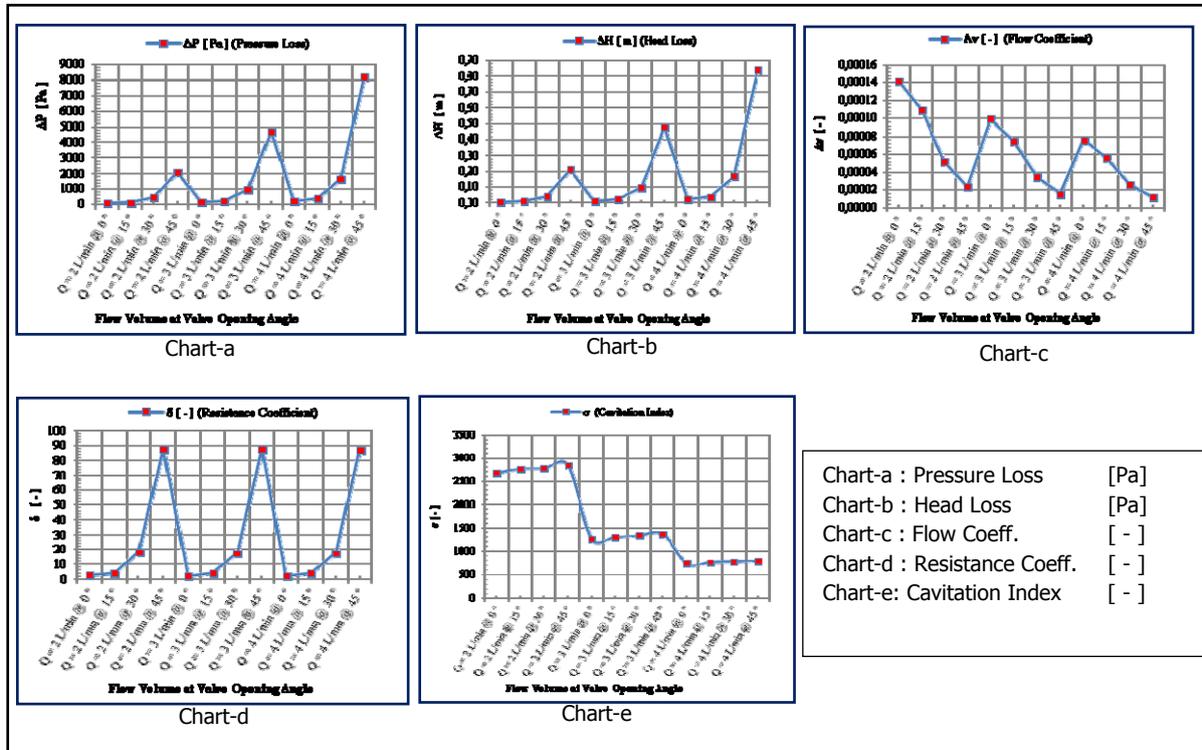


Figure 6. Comparison of flow parameters

CONCLUSIONS

Advanced CFD simulation applications in the field of agricultural engineering design are becoming ever more important due to the significant advantages that can be derived from its utilisation. Numerical simulations are a very powerful tool when used correctly. However, complementary experimental studies to validate the simulations are also essential to gain confidence in the results of such complicated numerical simulations; the results from such simulations will provide valuable information. CFD enables designers the opportunity to understand how geometrical and installation changes might impact irrigation equipment performance such as is the case with the mini valves study presented here. This will help the designer to innovate and optimise the product in any redesign that has to be undertaken.

In this paper, a case study using CFD techniques is presented for the prediction of several flow parameters for an irrigation mini valve. The flow parameters, which were used for related calculations in this paper based on CFD analysis results, were head loss, flow coefficient, resistance coefficient and

cavitation index. According to the results of CFD analyses, it is observed that the pressure loss changes with inlet flow volume rate and valve opening positions. If valve opening angle and inlet flow volume rates are increased, pressure loss, head loss and resistance coefficient are increased but flow coefficient is decreased. The cavitation indexes are generally high and cavitation possibility is not seen. Changes in flow parameters were also evaluated locally for the valve assembly. As one of the main points, it can be said that this study improved the understanding of the fluid performance of this particular irrigation valve, and as such, the method can also be applied for other agricultural irrigation equipment design evaluations.

In addition to all conclusions presented in the paper, leakage due to high value sudden pressures, manufacturing or material errors is another problem for this type of plastic mini valve. It may therefore be necessary to evaluate structural deflection / deformation cases for optimum part thickness of the valves to avoid leakage during its operation.

REFERENCES

- Baldygaa, J., Makowska, L., Orciucha, W., Sauterb, C., H., P., Schuchmannb, 2009. Agglomerate Dispersion in Cavitating Flows. *Chemical Engineering Research and Design*, Vol. 87(4): 474-484
- Chern, M-J., Wang, C-C., C-H., Ma, 2007. Performance Test and Flow Visualization of Ball Valve. *Experimental Thermal and Fluid Science*, Vol. 31: 505-512.
- Falvey, H., T., 1990. Cavitation in Chutes and Spillways. *Water Resources Technical Publication*, Engineering Monograph No.42, Denver, USA
- Forsberg, J., T., Persson, 2005. CFD-tools in valve design. Project Report, Karlstad University, Division for Engineering Sciences, Physics and Mathematics, Department of Environmental and Energy Systems. Available from URL: <http://www.uppdragsutbildningen.se/om/pdf/Delprojekt%206.pdf>, [Last accessed: 15.01.2011].
- Ivanell, S., 2001. Hydrodynamic Simulation of a Torpedo with Pump Jet Propulsion System. MSc Thesis, Engineering Physics Dept., Royal Institute of Technology, Stockholm, Sweden.
- Kabwe, A.M., Fester, V.G., P.T., Slatter, 2010. Prediction of Non-Newtonian Head Losses through Diaphragm Valves at Different Opening Positions. *Chemical Engineering Research and Design*, Vol. 88: 959-970.
- Lee, J-H., Song, X-G., Park, Y-C., S-M., Kang, 2010a. Computational Fluid Dynamic Analysis of Flow Coefficient for Pan Check Valve. *Proceedings of the 9th WSEAS Intl. Conference on Applied Computer and Applied Computational Science*, pp.157-160, ISBN: 978-960-474-173-1.
- Lee, J-H., Song, X-G., Kang, S-M., Y-C., Park, 2010b. Optimization of Flow Coefficient for Pan Check Valve by Fluid Dynamic Analysis. *Proceedings of the 3rd Global Conference on Power Control and Optimization. AIP Conference Proceedings*, Vol. 1239: 337-340.
- Lomax, H., Pulliam, T., H., Zingg, D., W. 1999, *Fundamentals of Computational Fluid Dynamics*. NASA Ames Research Centre and University of Toronto Institute for Aerospace Studies. Available from URL: http://maji.utsi.edu/courses/07_681_advanced_viscous_flow/ref_af6 *Fundamentals of CFD*. pdf, [Last accessed: 03.02.2011]
- Min, B., Xin, F., C., Ying, 2008. Computational Fluid Dynamics Approach to Pressure Loss Analysis of Hydraulic Spool Valve. *Technical University of Karlsruhe (TH), Germany*, Available from URL: <http://citeseerx.ist.psu.edu/viewdoc/download?doi=10.1.1.111.6440&rep=rep1&type=pdf>, [Last accessed: 01.02.2011].
- Mokhtarzadeh-Dehghan, M., R., Ladommatos N., T.J., Brennan, 1997. *Finite Element Analysis of Flow in A Hydraulic Pressure Valve*. *Applied Mathematical Modelling* Vol. 21 (7): 437-445.

- Nicholls, R.F., S.R., Turnock, 2007. The Use of Computational Fluid Dynamics in the Optimisation of Marine Current Turbines. NUTTS 2007:1-6, Available from URL: http://eprints.soton.ac.uk/48787/1/NuTTS_2007_RNL.pdf, [Last accessed: 11.01.2011].
- Prakash, K.S., Nazirudeen, S.S.M., Malvinraj, M.J., T., Manohar, 2009. Integration of Material Design and Product Design: A CFD Based Approach, International Journal of Dynamics of Fluids, Vol. 5 (2): 199-214.
- Schiavello, B., F.C., Visser, 2008. Pump Cavitation, Various NPSHR Criteria, NPSHA Margins and Impeller Life Expectancy. The 24th International Pump Users Symposium, Available from URL: <http://turbolab.tamu.edu/uploads/files/papers/p23/P23Tut04.pdf>, [Last accessed: 11.02.2011]
- Smith, P., R.W., Zappe, 2004. Valve Selection Handbook - 5th Edition. Elsevier, Inc. Press, ISBN: 0-7506-7717-1, USA.
- SolidWorks Product. 2010a. Determination of Hydraulic Loss (Section 4), SolidWorks Flow Simulation Online Tutorial.
- Solidworks Product.2010b. Flow Simulation Online User's Guide File.
- Song, X.G., Y.C., Park, 2007. Numerical Analysis of Butterfly Valve-Prediction of Flow Coefficient and Hydrodynamic Torque Coefficient. Proceedings of the World Congress on Engineering and Computer Science 2007 (WCECS 2007), October 24-26, San Francisco, USA.
- Song, X-G., Kim, S-G., Baek, S-H, Y-C., Park, 2009. Structural Optimization for Ball Valve Made of CF8M Stainless Steel. Trans. Nonferrous Met. Soc. China Vol. 19: 258–261.
- Recktenwald, G., 2007. Head Loss in Pipe Flow Major and Minor Losses. ME322 Lecture Slides, Mechanical and Materials Engineering Dept., Portland State University, Portland, USA., Available from URL: http://web.cecs.pdx.edu/~gerry/class/ME322/notes/pdf/ME322_lect04slides.pdf, [Last accessed: 08.02.2011]
- Val-Matic. 2010. Cavitation in Valves. Val-Matic Valve and Manufacturing Corp. USA, Available from URL: www.valmatic.com, [Last accessed: 10.02.2011].