Springer

Journal of Mechanical Science and Technology 22 (2008) 1998~2005

Journal of
Mechanical
Science and
Technology

www.springerlink.com/content/1738-494x
DOI 10.1007/s12206-008-0745-6

Numerical analysis of 3-D flow through LNG marine control valves for their advanced design†

Young Joon An, Byeong Jin Kim and Byeong Rog Shin*

Department of Mechanical Engineering, Changwon National University, 9 Sarim-dong, Changwon 641-773, Korea

(Manuscript Received April 16, 2008; Revised July 11, 2008; Accepted July 23, 2008)

Abstract

A numerical analysis of three dimensional incompressible turbulent flows through high pressure drop control valves was carried out by using a CFD-ACE code to develop anti-cavitation control valve used in LNG marine system. For this, numerical simulation was performed on several models of control valve that have different orifice diameters of anti-trim and the size of valve discharge. In this study, flow characteristics of control valves with complex flow fields including pressure drop, cavitation effect and variation of flow coefficient as well as correlation of discharge coefficient were investigated and analyzed. Comparing with conventional control valves, newly designed valves by using the CFD analysis showed an improved flow pattern with reduced cavitation and an anticipated performance characteristic.

Keywords: CFD analysis; Cavitating flow; Control valve; Anti-cavitation trim; LNG marines

---

1. Introduction

Recently with the world wide interest in the use of natural gas, the demand of transportation by using LNG marine system is shown a sudden increment. In this system, high temperature and high pressure control valves which are one of the most important fluid devices are widely used safely to control storage and delivery of the liquefied LNG. Unfortunately, however, during the operation cavitation is occurred frequently and brings an unexpected damage to valve systems.

Up to the date, in order to control the flow-rate, pressure and flow directions, many kinds of valves such as glove valve, gate valve, butterfly valve and ball valve are used at industry. Among these, high pressure drop control valve in adjust pressure differences in LNG marines is a sort of glove valve. In this valve, due to the rapid change of flow rate with high pressure drop in accordance with operating- and design conditions, the flow frequently accompanies cavitation. Cavitation is a phase change phenomenon occurred in some domain below vapor pressure according to the decrease of local pressure when fluid devices like a LNG marine move with a high-speed in working fluid of liquid state. When cavitation occurs unexpectedly accompanying the attachment and collapsing of cavitation bubbles near solid surfaces, it causes the noise, vibration and damage as well as changes the performance characteristics in hydraulic systems[1]. As the results, it has an unfavorable effect on the performance and eventually brings the low efficiency. In the sense of reducing these in the valve system, therefore, technology of accurate prediction and estimation of cavitation are very important in advanced design and development of high pressure drop control valves.

In the cavitating flow analysis, numerical approach is advantageous because cavity flow includes complex and strong unsteady phenomena with phase changes, fluid transients, vortex shedding and turbulence. Unfortunately, however, because of compli-

---

† This paper was presented at the 9th Asian International Conference on Fluid Machinery (AICFM9), Jeju, Korea, October 16-19, 2007.

*Corresponding author. Tel.: +82 55 213 3609, Fax.: +82 55 275 0101
E-mail address: brshin@changwon.ac.kr

© KSME & Springer 2008

cated geometry of glove type high pressure control valve, numerical analysis is very difficult. Recently, Chung et al.[2] carried out numerical analysis of three dimensional (3-D) flow in glove valves according to the change of the valve opening by using FLUENT commercial code, and valve flow characteristics such as pressure drop and flow rate coefficient were estimated and compared with experimental data. Yoon et al.[3] also simulated 3-D turbulent flow with structured grid system and investigated pressure distributions in glove valve with valve trim to check the occurrence of cavitation by using CFD-ACE commercial code.

In this paper, to develop anti-cavitation control valve using in LNG marine system, a numerical analysis of 3-D incompressible turbulent flows through high pressure drop control valves is carried out by using the CFD-ACE[4]. For this, numerical simulation is performed at several valve models both newly designed and conventional control valve that have different orifice diameters of anti-trim and the size of valve discharge. And then, flow characteristics of control valves with complex flow fields including pressure drop, cavitation effect and variation of flow coefficient as well as correlation of discharge coefficient are investigated and analyzed. Comparing with conventional control valves, the newly designed valves by using present CFD analysis to reduce cavitation showed an improved flow pattern with reduced cavitation and an anticipated performance characteristic.

2. Fundamental equations

In this study, cavitating flows through control valves are simulated by using multi-blocked unstructured grid system for real type valve shown in Fig. 1 to capture accurate 3-D flow phenomena. The simulation was performed for various shapes of valve trim and number of trim holes.

The fundamental equations are Reynolds averaged Navier-Stokes equations using standard $\kappa-\varepsilon$ turbulence model for solving turbulence flow. Governing equations and turbulence transport equations for the eddy kinetic energy κ and the turbulence dispassion ε are written as follows:

$$\begin{aligned}&\frac{\partial}{\partial t}\left(\rho u_j\right)+\frac{\partial}{\partial x_j}\left(\rho u_i u_j\right)\\&=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\rho\overline{u_i' u_j'}\right)\right]\end{aligned}\tag{1}$$

$$\begin{aligned}&\frac{\partial}{\partial t}\left(\rho\kappa\right)+\frac{\partial}{\partial x_j}\left(\rho u_j\kappa\right)\\&=\rho P-\rho\varepsilon+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\kappa}\right)\frac{\partial\kappa}{\partial x_j}\right]\end{aligned}\tag{2}$$

$$\begin{aligned}&\frac{\partial}{\partial t}\left(\rho\varepsilon\right)+\frac{\partial}{\partial x_j}\left(\rho u_j\varepsilon\right)\\&=C_{\varepsilon_1}\frac{\rho P\varepsilon}{\kappa}-C_{\varepsilon_{12}}\frac{\rho\varepsilon^2}{\kappa}+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\kappa}\right)\frac{\partial\varepsilon}{\partial x_j}\right]\end{aligned}\tag{3}$$

where, $\overline{u_i'u_j'}=(2/3)k\delta_{ij}-\nu_t(\partial u_i/\partial x_j+\partial u_j/\partial x_i)$. Eddy viscosity μ_t and the production term P is given by

$$\mu_t=\rho C_\mu\frac{\kappa^2}{\varepsilon}\tag{4}$$

$$P=\nu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\frac{\partial u_m}{\partial x_m}\delta_{ij}\right)\frac{\partial u_i}{\partial x_j}-\frac{2}{3}\kappa\frac{\partial u_m}{\partial x_m}\tag{5}$$

Also, the transport equation for vapor of cavitation is as follow with effective exchange coefficient:

$$\frac{\partial}{\partial t}\left(\rho f\right)+\nabla\cdot\left(\rho\vec{V}f\right)=\nabla\cdot\left(\Gamma\nabla f\right)+R_e+R_c\tag{6}$$

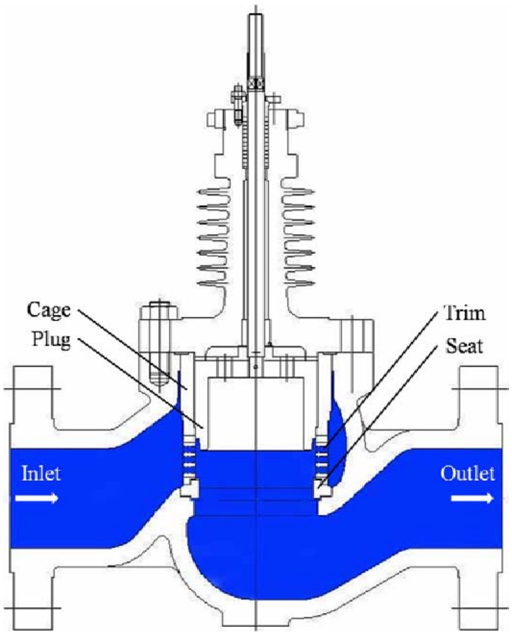


Fig. 1. Schematic diagram of control valve.

where, ρ is mixture density and f is vapor mass fractions. The relationship between ρ and f is given as

$$\frac{1}{\rho} \equiv \frac{f}{\rho_v} + \frac{1-f}{\rho_l} \tag{7}$$

where, ρ_v and ρ_l represent the vapor and liquid densities, respectively. R_e and R_c are source terms to consider the phase change for evaporation and condensation in vapor transport Eq. (6).

$$R_e = C_e \frac{V_{ch}}{\sigma} \rho_l \rho_v \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_l}} (1-f) \tag{8}$$

$$R_c = C_c \frac{V_{ch}}{\sigma} \rho_l \rho_v \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_l}} f \tag{9}$$

where, p_v is vapor pressure. C_e and C_c are empirical constants. σ is surface tension at interface. Characteristic coefficient V_{ch} is approximated by the local turbulence intensity and given as $V_{ch} = \sqrt{\kappa}$ [4].

3. Computational grid

Computational grid for both conventional control valve with a wedge type valve trim and the newly designed control valves with cylindrical orifice type anti-cavitation trim for effective control of cavitation are generated by using unstructured multi-block grids consisted of prism, tetrahedron and hexahedron elements. The valve openings are 40%, 60%, 80% and 100%.

Fig. 2 shows flow passages and computational grids for conventional and newly designed 200A valve with 162 orifices. Mesh density is high around valve trim and near wall.

4. Numerical method

A SIMPLEC algorithm based on finite volume method in the CFD-ACE program is used to analyze incompressible flow in the valves. The working fluid was assumed as a pure water with air content of 1.5×10^{-5} at 20℃. The inlet and outlet total pressure of 6.205MPa and 1.013MPa are imposed. In this study, total 5 models such as conventional type valve model, the newly designed valves models of 200A, 150A, 50A and 32A classified by the valve size and shape are computed. Among the models, 200A is the improved model of conventional one. By CFD analysis orifice whole instead of wedge type whole of conventional valve as trim whole is adopted in the newly designed valves, and it reduces the occurrence of cavitation.

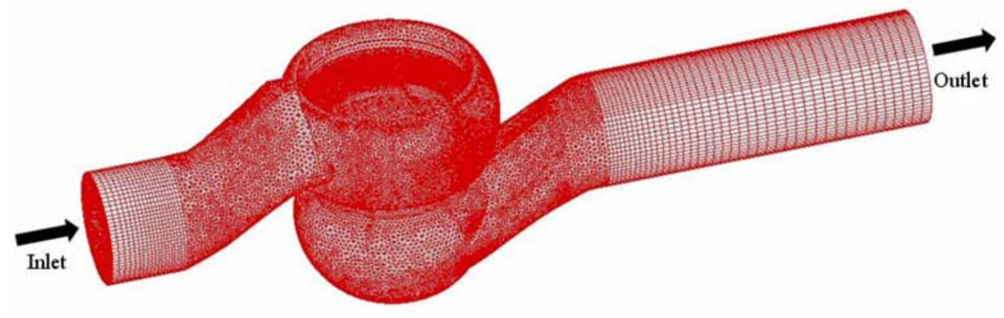
5. Numerical results

Fig. 3 shows pressure distribution for several valves models on symmetry plane at 80% valve opening.

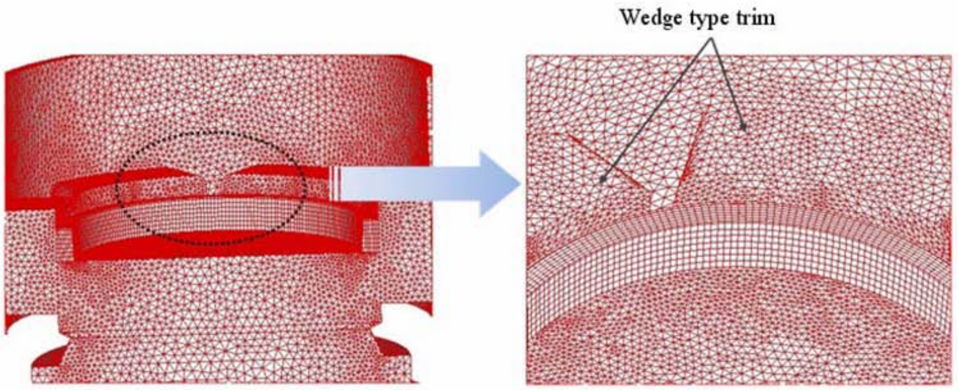
Comparing with conventional model with wedge type trim holes (Fig. 3 (a)), it is show that the newly designed valve with cylindrical trim holes in Fig. 3(b) indicates somewhat low pressure distribution in the trim passage. But it is still over the vapor pressure. In general, at the expanded pipe regions, relatively low pressure is distributed like a sudden expansion pipe flow. Fig. 4 shows a pressure drop along the center line of valve passage for several control valves. In the front of valve trim, pressure is increased by flow impinging due to the sudden contraction of flow passage. After the trim, with the big change of flow passage in valve chamber large pressure drop is shown. The lowest pressure in 200A valve is higher than that of conventional one. It means the possible cause of occurrence of cavitation in new design valve is rare. This improvement is much clearer at the lower wall in Fig. 5.

Fig. 6 shows velocity vectors and streamlines for 200 type valves. It is well simulated that the flow incoming through the trims flows toward exit by way of valve chamber. In the case of conventional type valve, unstable and relatively high velocity region, and large circulation zone exist at valve chamber because of large inlet area of wedge type trim. While, in the improved 200A the flow velocity is somewhat fast at the trim because of a narrow path. Then it is recovered with uniform distribution at the valve chamber, so that the circulation zone is smaller than that of conventional type.

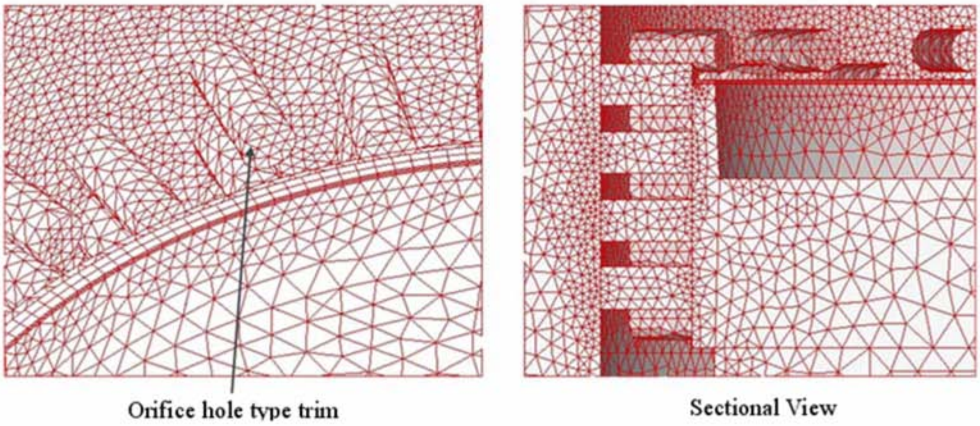
Fig. 7 shows computed flow patterns of pressure, void fraction contours and streamlines with velocity vectors on cross-sectional plane of near trim center at 80% valve opening. Conventional type shows irregular velocities between the trim and the chamber. Especially, relatively low pressure and high void fraction existing cavitaton is distributed at the entrance of the trim. In the newly designed 200A model, the flow incoming from the circumferential direction is well



(a) Overall view

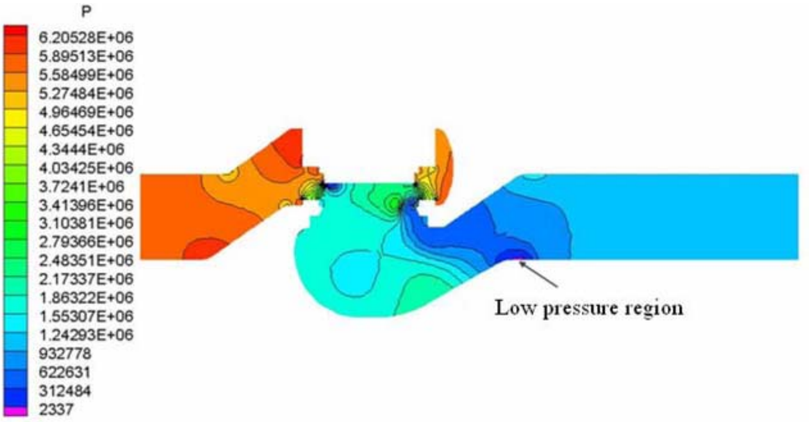


(b) Near trim (conventional type)

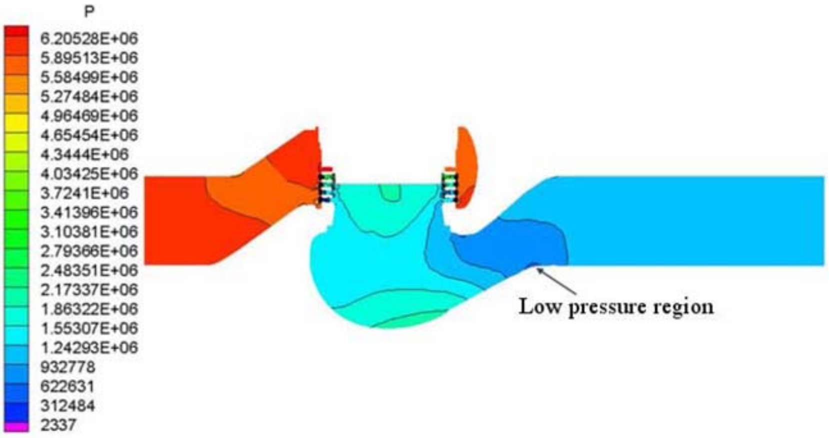


(c) Near trim (newly designed, 200A)

Fig. 2. Computational grid of 200 type valve.



(a) Conventional valve



(b) 200A valve

Fig. 3. Pressure contours of 200 type valves.

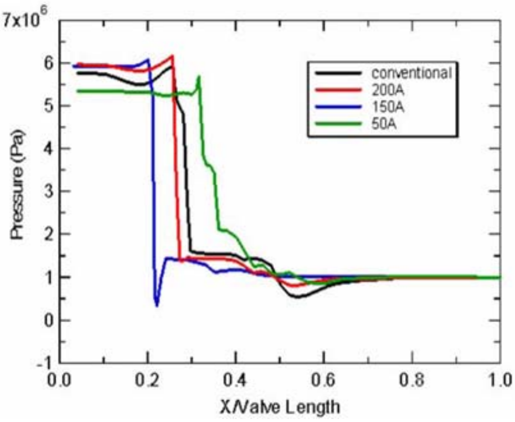


Fig. 4. Pressure drop along the valve centerline at 80% valve opening.

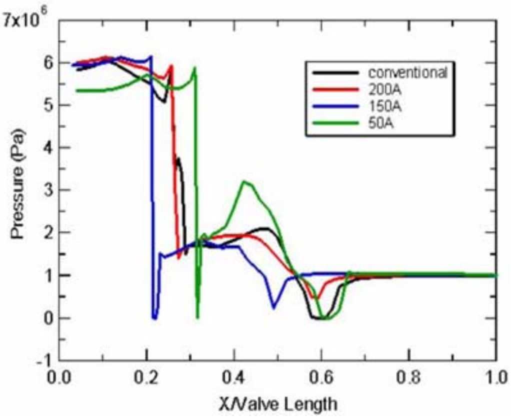
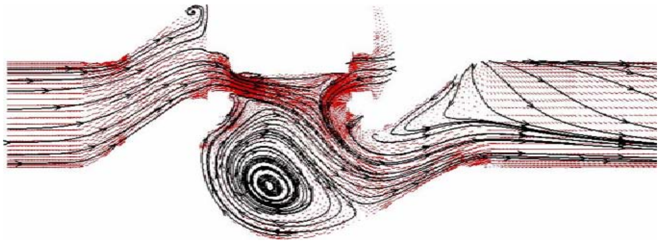
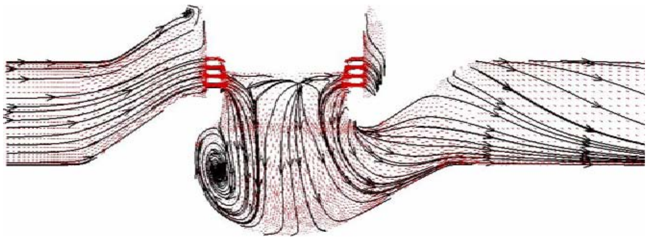


Fig. 5. Pressure drop along the lower wall at 80% valve opening.

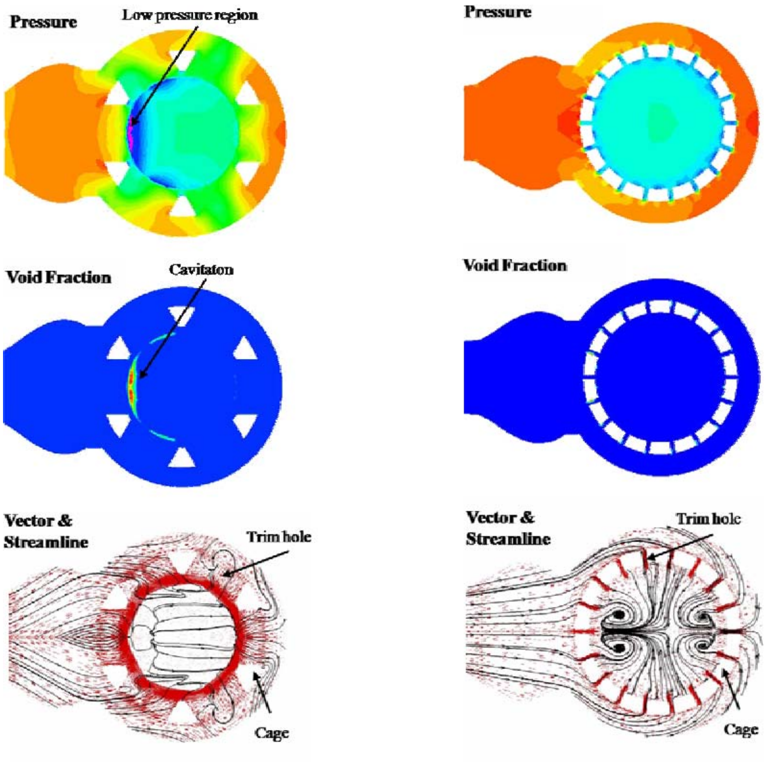


(a) Conventional valve



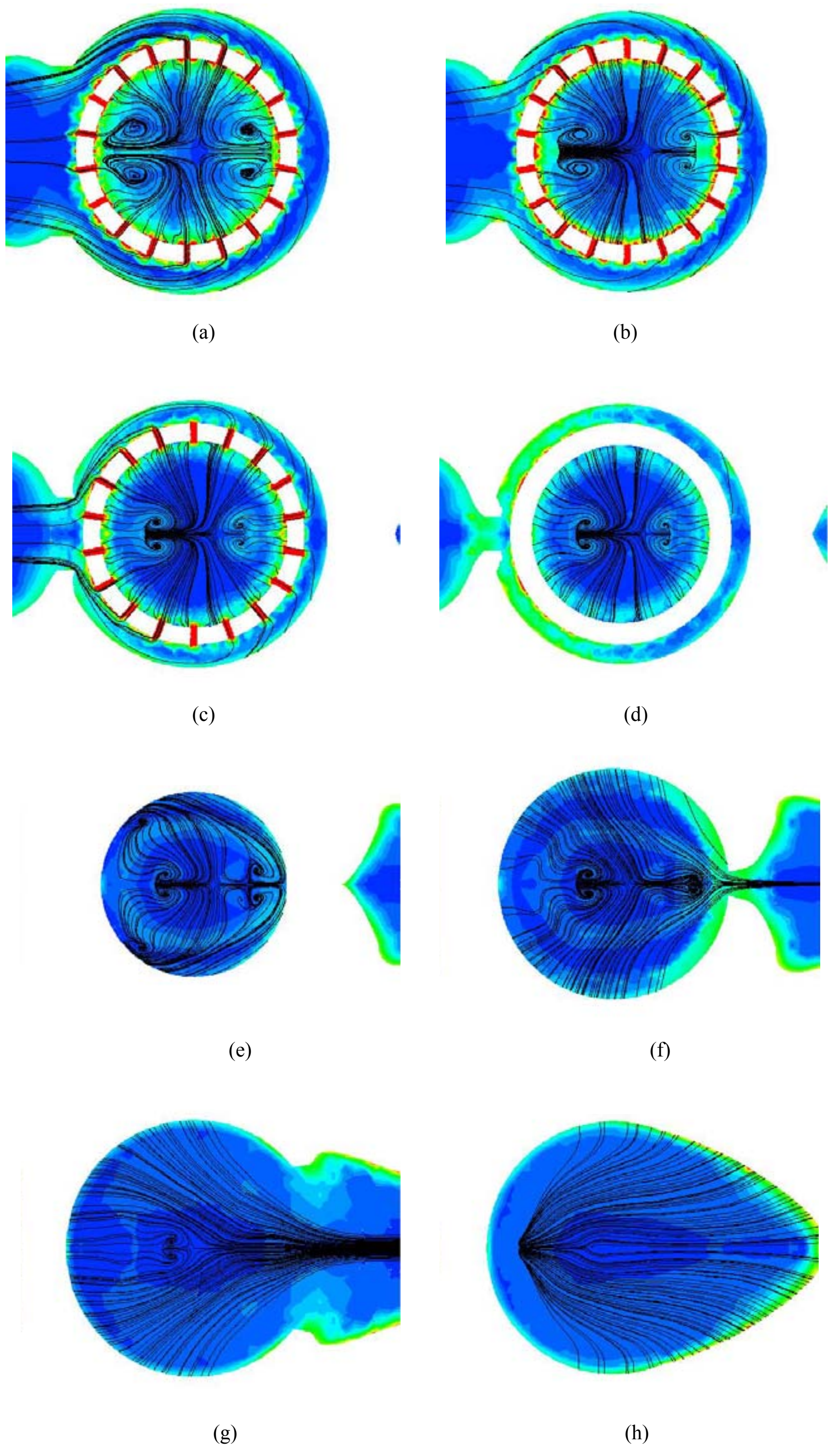
(b) 200A valve

Fig. 6. Velocity vector and streamlines at 200 type valve.



(a) Conventional valve (b) 200A valve

Fig. 7. Flow patterns of pressure, void fraction contours and streamline with velocity vectors on cross-sectional plane at 80% valve opening.



(a) (b)

(c) (d)

(e) (f)

(g) (h)

Fig. 8. Vorticity magnitude contours and streamlines at several cross sections from top to bottom of 200A valve.

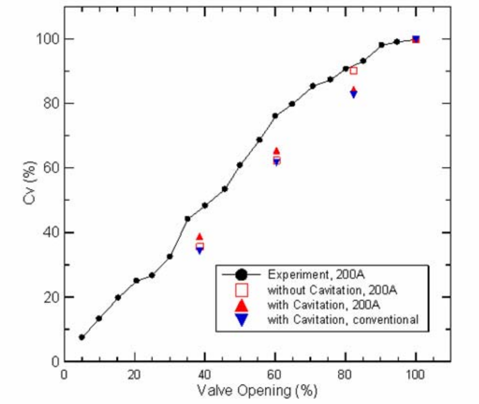


Fig. 9. Valve flow coefficient.

distributed to 18 trim holes and flows into the chamber with two vortex pairs and it is flowing toward axial direction as shown in Fig. 8. Pressure distribution is quite good at the whole regions. No cavitation is observed.

On the other hand, flow coefficient which is one of characteristics of the valve is used to determine valve diameters and estimated by using $C_V = 1.17Q\sqrt{G/\Delta p}$, where, Q is flow rate, G is specific gravity and △p is pressure drop between inlet and outlet of the valve.

Fig. 9 shows a comparison of the flow coefficient with experimental data for the 200A valve at several valve opening. As a whole, the present result has a good tendency with experimental data[5]. The results obtained by present simulation with cavitation effect are better than that without consideration of cavitation. 200A is closer to the experiments than the conventional one.

6. Conclusion

A 3-D incompressible cavitating flow was simulated to investigate internal flow characteristic in high pressure drop control valve with anti-cavitation trim for the LNG marine by using the CFD-ACE. Overall, flows through the valve including the trim region, valve chamber, valve inlet and outlet are well simulated, so that it is easy to understand complicated flow phenomena in valve system. The newly designed 200A valve to reduce cavitation shows a rectified velocity and pressure distribution with no cavitation at the whole valve areas comparing with conventional one. It shows a good comparison of the flow coefficient with experimental data at several openings for 200 type valve. 200A is better than the conventional one resulting superior function of anti-cavitation trim.

Acknowledgements

A part of this research is financially supported by Underwater Vehicle Research Center (UVRC), DAPA of Korea.

References

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

[2] M. H. Chung, S. M. Yang and H. Y. Lee, 3D flow analysis of globe valve with air operated actuator, *Journal of Fluid Machinery*. 8 (2005) 7-13.

[3] J. Y. Yoon, S. J. Byun, J. M. Yang and D. H. Lee, Numerical analysis of the 3-D flow field in a globe valve trim under high pressure drop, *Journal of Fluid Machinery*. 4 (2001) 14-20.

[4] ESI-Group, CFD-ACE User Manual, Version 2006, ESI-Group (2006).

[5] H. S. Noh and B. J. Kim, Research Report of the Small and Medium Business Administration on Development of High Pressure Drop Control Valve for LNG Marine, BY Controls Co, (2006).